TritonUAS Solidworks Workshop 2023-2024

Welcome to the TritonUAS Solidworks Workshop! Here we will be describing the basics on how to use the program and some practice models. The layout of the workshop goes as follows:

1. Software Introduction
2. Features/Practice models
    a. Block
    b. Button
    c. Wing

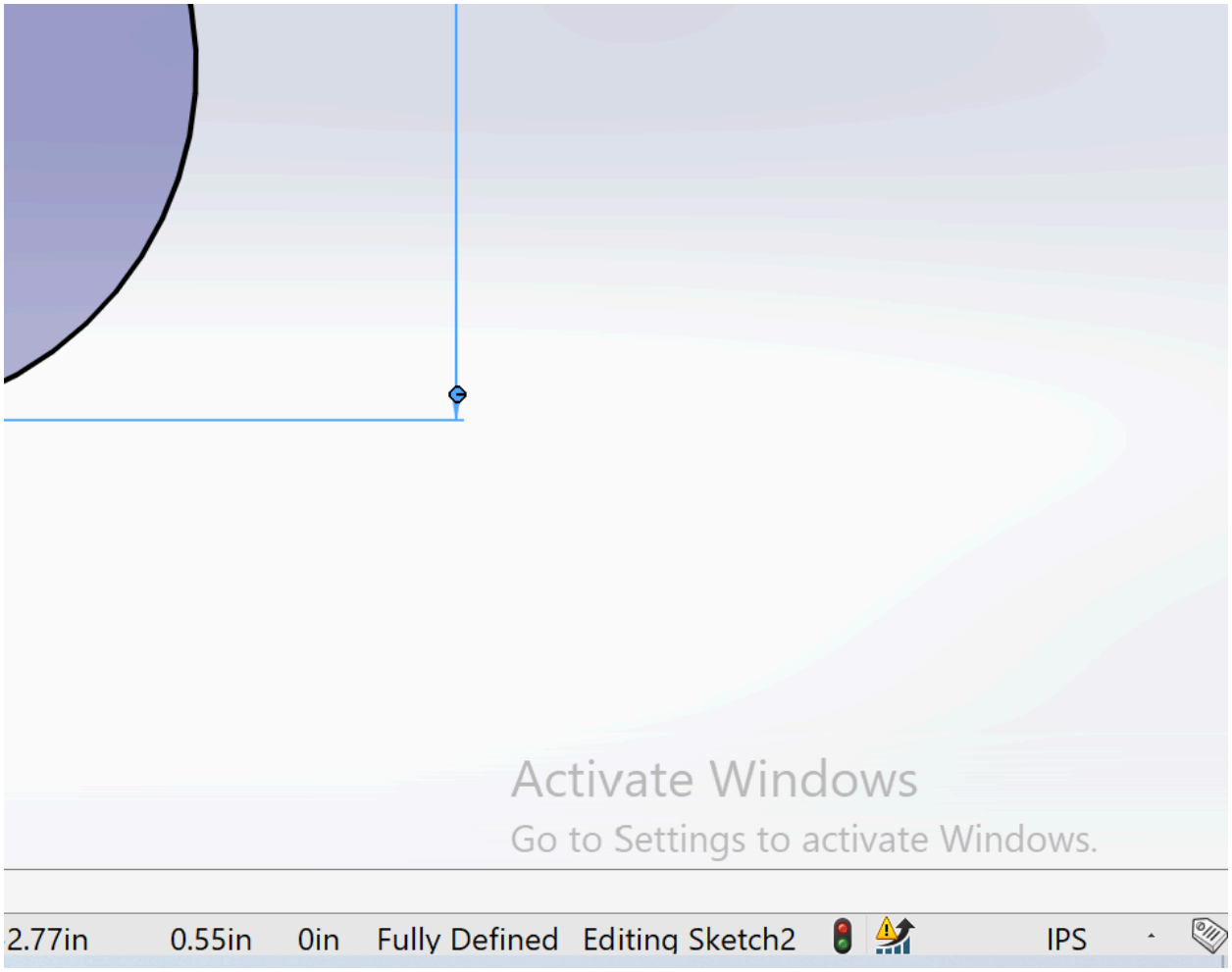
Software Introduction

Solidworks is a useful 3D modeling computer aided design (CAD) software that is used prominently as a design and prototyping tool. You can use it to create a virtual model of what you want to make to see if it will function properly instead of constant trial and error, which can be costly in terms of material and time. There are other functions that could be used like a stress simulator, but there are other softwares like ABAQUS and ANSYS that are more suited for those tasks. Solidworks can be tricky and unintuitive, so this tutorial will go over practical functions that we use in our organization to help get you started. We will also give a few examples of things we can make using these functions. We recommend you use a mouse, you can use the trackpad but it will be painful.

Features

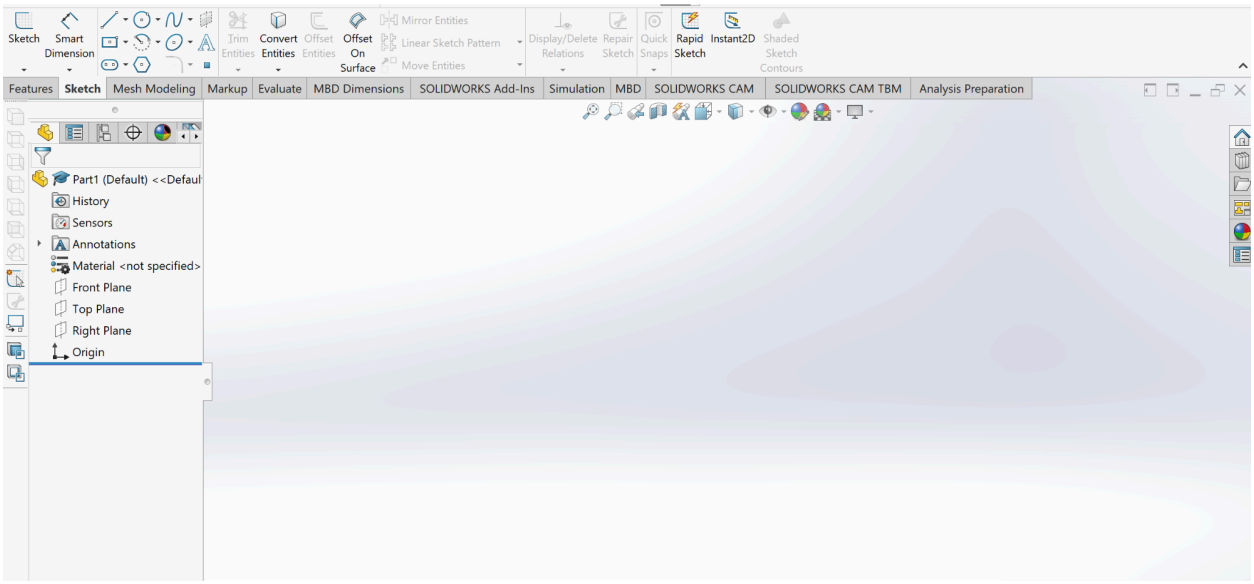
Here are some of the most used features we use in TritonUAS.

Starting from when you first open Solidworks, an important habit to develop is to make sure you are in the right units. In the bottom right corner there are letters. It looks like this:



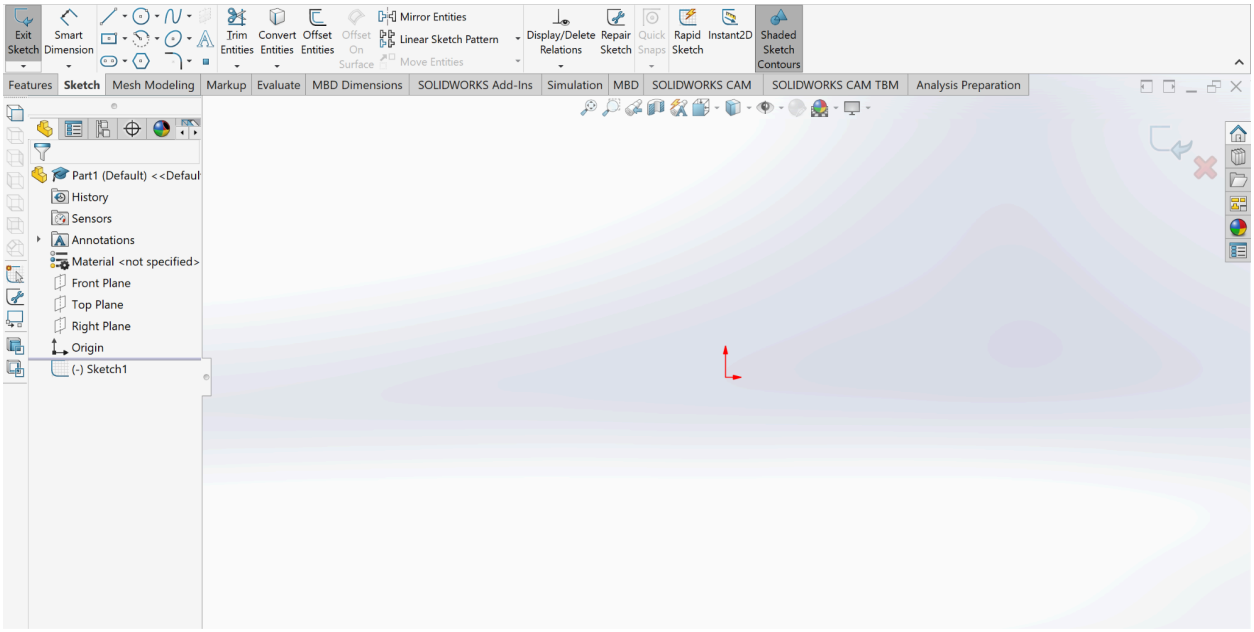
The IPS is what we are looking at. If you click it, you get four options for units. It will explain to us what they are representing. Always look here first to make sure you are working with the correct units because it will be very confusing when your dimensions are all wrong when you laser cut (This has happened too many times to me).

Now let's get into the fun stuff: modeling. Look at the top ribbon and click on 'Sketch'. Sketching is the start of the model as you start in 2D before extruding into 3D.

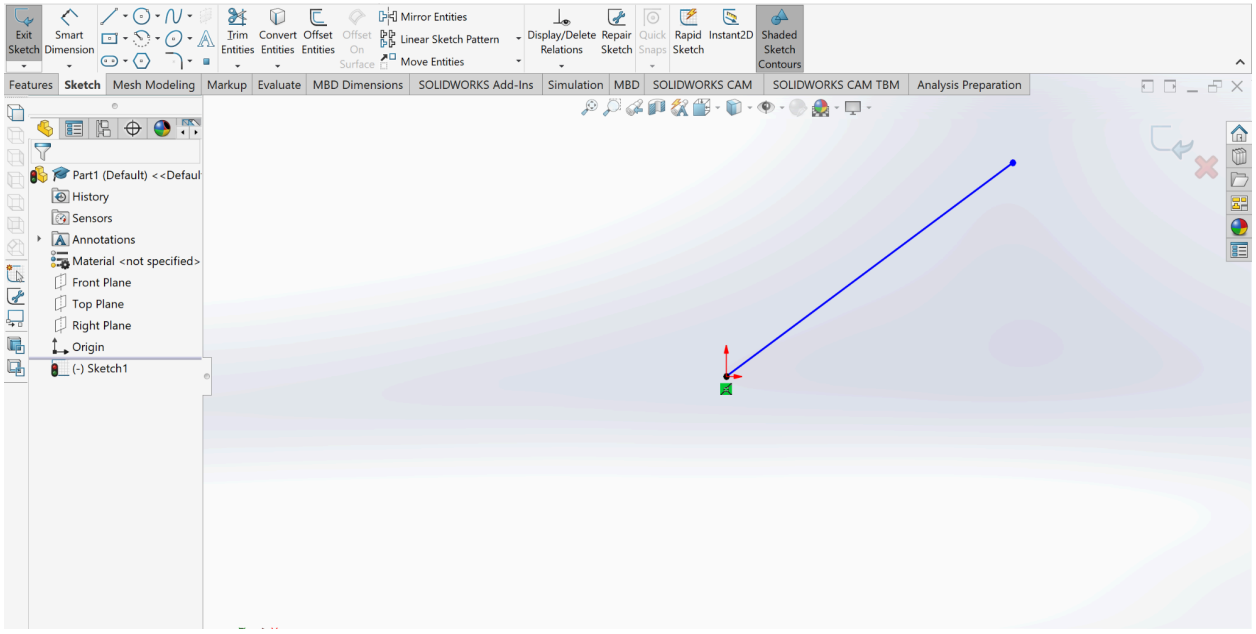


When you select 'Sketch', there are three planes you can draw on: Top, Right, and Front. You can click anyone to draw on, it really only changes the axis you draw on or set as your "bottom", but in this example, I will click Top.

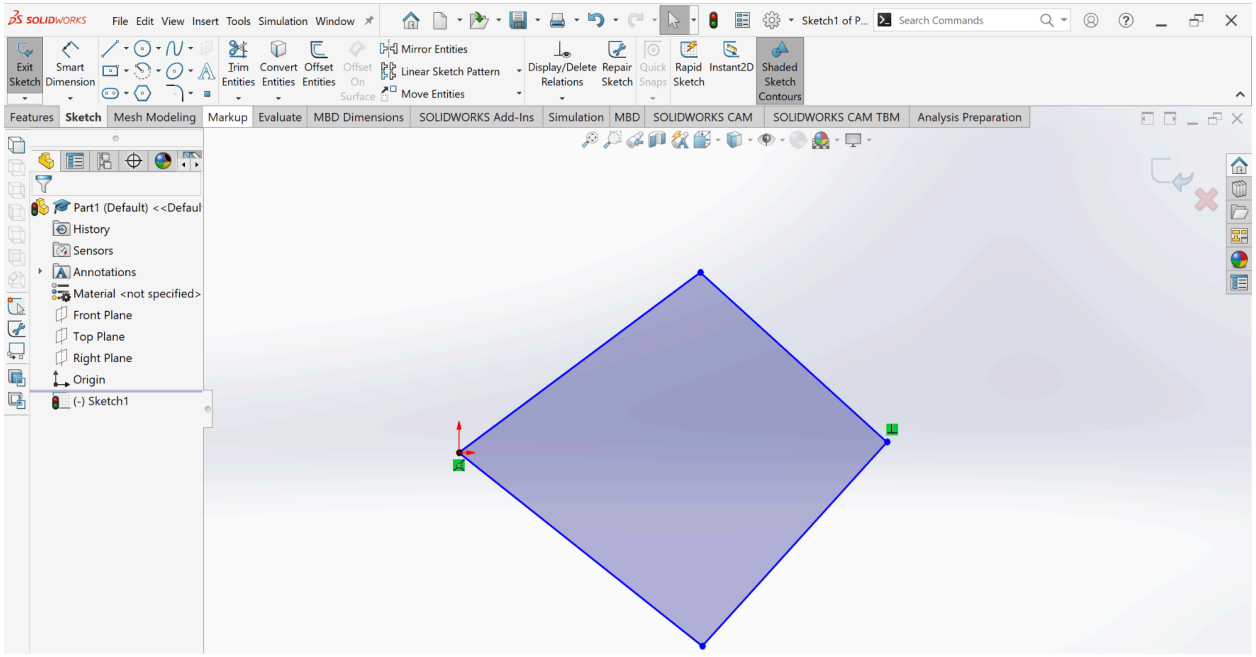
Next to the Smart Dimension button, there are a lot of buttons we can press.



Let's go over a few of them. The top left on is the line command. By clicking the drop down next to the line command, you can select line, centerline, or midpoint line. The normal line works by clicking anywhere on the plane to start the line, moving the mouse to extend the line, and clicking again on where you want the line to end. Centerline works the same way however this is mainly for dimensioning or referencing, as the construction line has no tangible effect on the model. Midpoint line is like the normal except when you click on the plane, the line starts from the middle and extends in both directions evenly. Moving the mouse and clicking again extends and ends the line respectively. It should look like this:

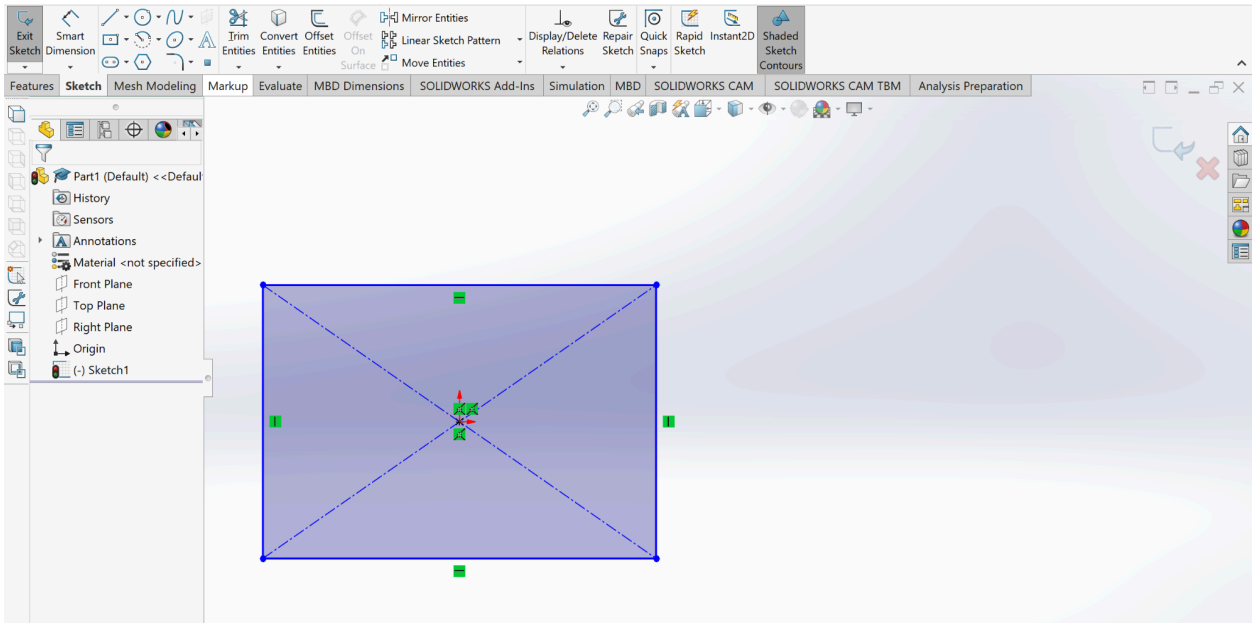


You can also continue this process and connect the lines together to make shapes.



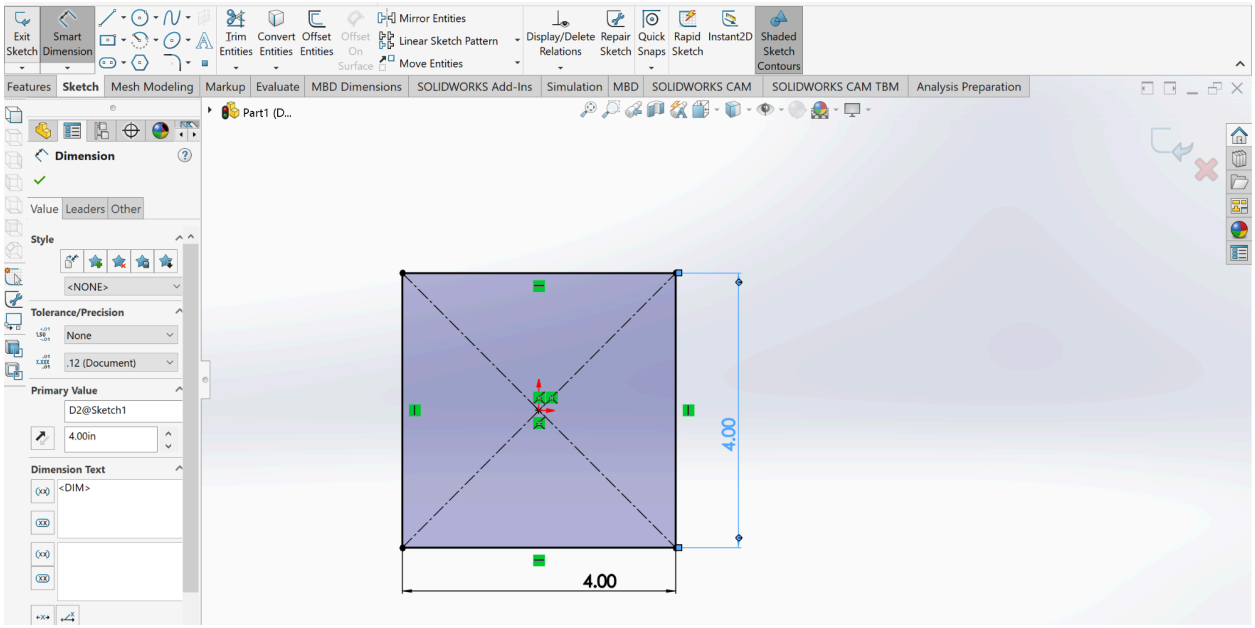
But wait, making shapes that way is not efficient. You can also make circles, squares, arcs, etc. using the other functions next to the line command. There are plenty of ways to make these shapes, which will depend on what situations you need to make your shape fit better.

As an example, let's make a solid square box. First, make sure you are sketching.The sketch button will say exit sketch if you are in sketch mode. Select the square/rectangle command right below the line command.
If you click the drop down, there are 5 different ways of making your rectangle. You can hover over them to see how they work. The one we are going to use is the center rectangle command which is the 2nd one from the top of the drop down. This allows you to start making a rectangle by selecting its center point, and then moving the mouse to extend the rectangle in the x and y direction. Clicking again creates the rectangle. In the image below, I selected the center of the rectangle to be at the origin of my plane, which is the intersection of the red arrows.



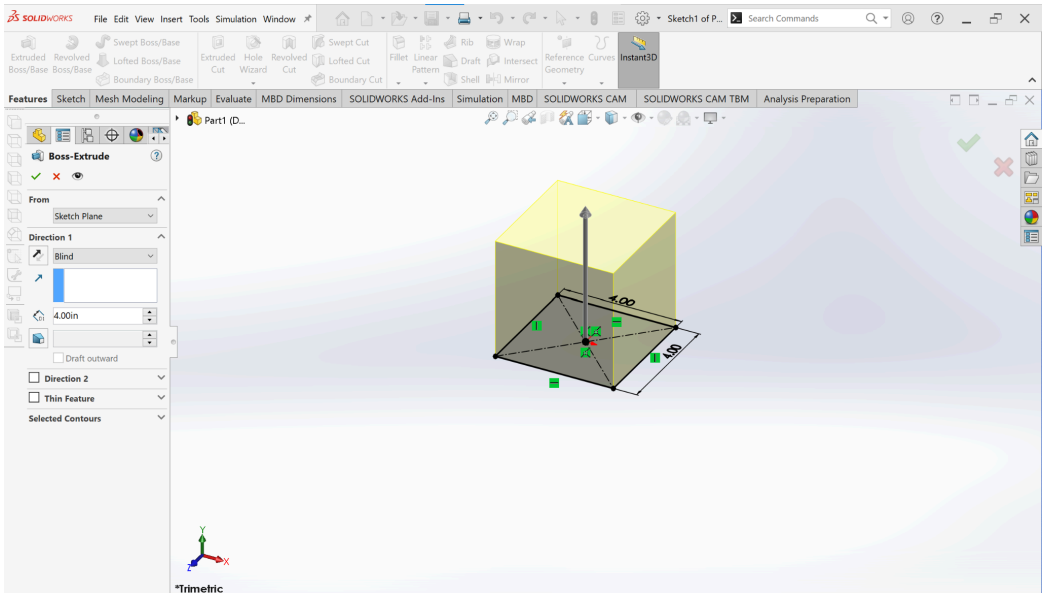
However, this may be a beautiful rectangle, we want a square, and I don't know about you, but the sides don't look equal. Let's fix that. Staying in the sketch, click the Smart Dimension command right next to the sketch command. This allows you to change the dimensions of

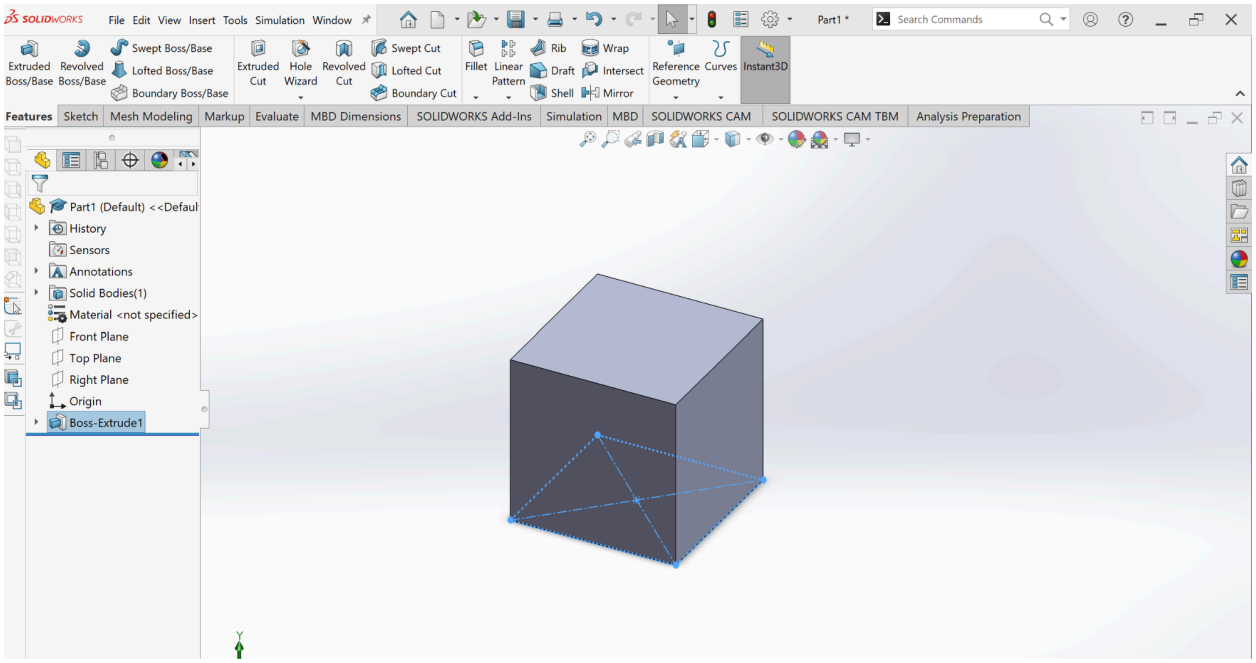
your lines to what you want exactly. For this example, the dimension of the square is 4 in by 4 in. It should look like below



You could also double click the numbers to change those dimensions.

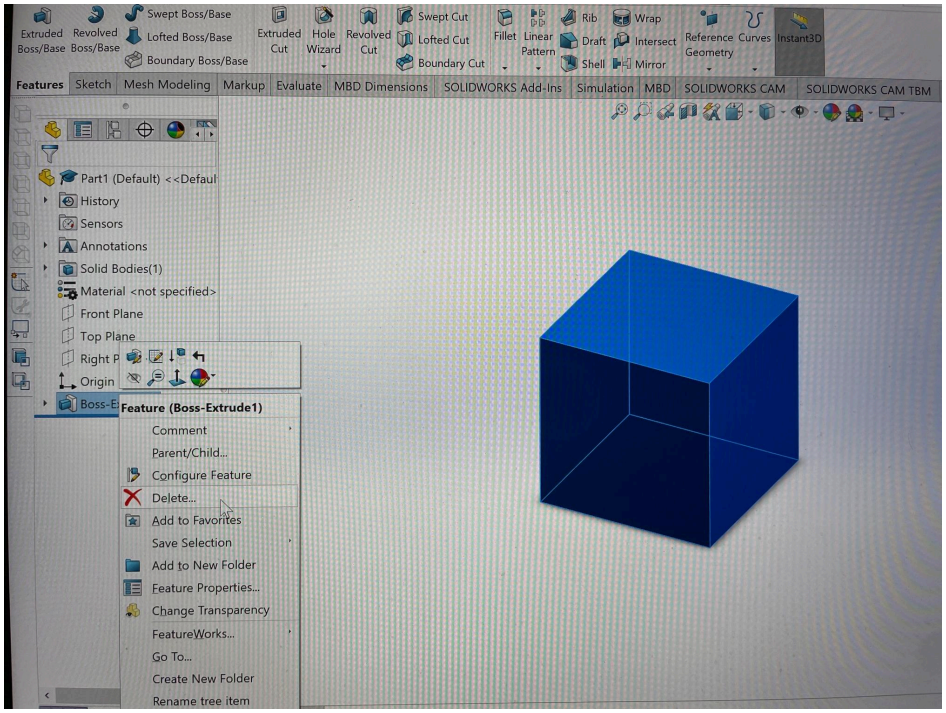
Finally, click on the Features tab next to sketch and hit boss-extrude. This turns your 2D drawing into a 3D shape. There are a bunch of ways you can extrude depending on the situation, but in this example, we will just stick with blind which allows you to control the extension by clicking and dragging. You can also type in how far you want to extrude by typing it in on the left underneath blind. It should look like this:



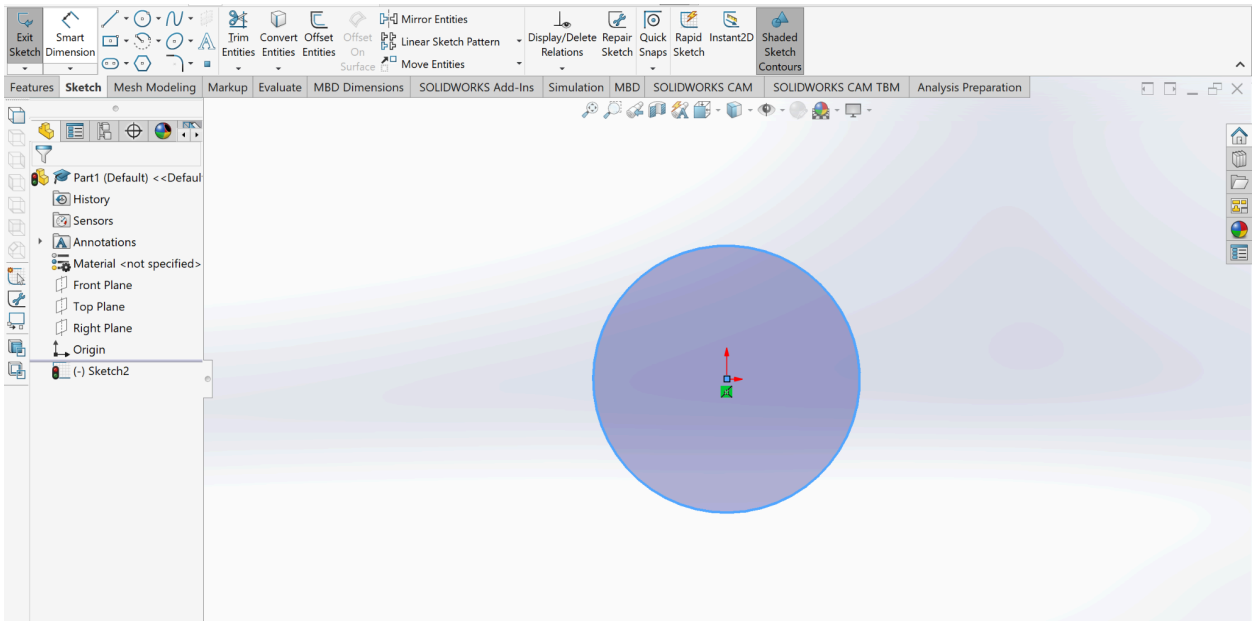


Congratulations! You have made your first solid model.

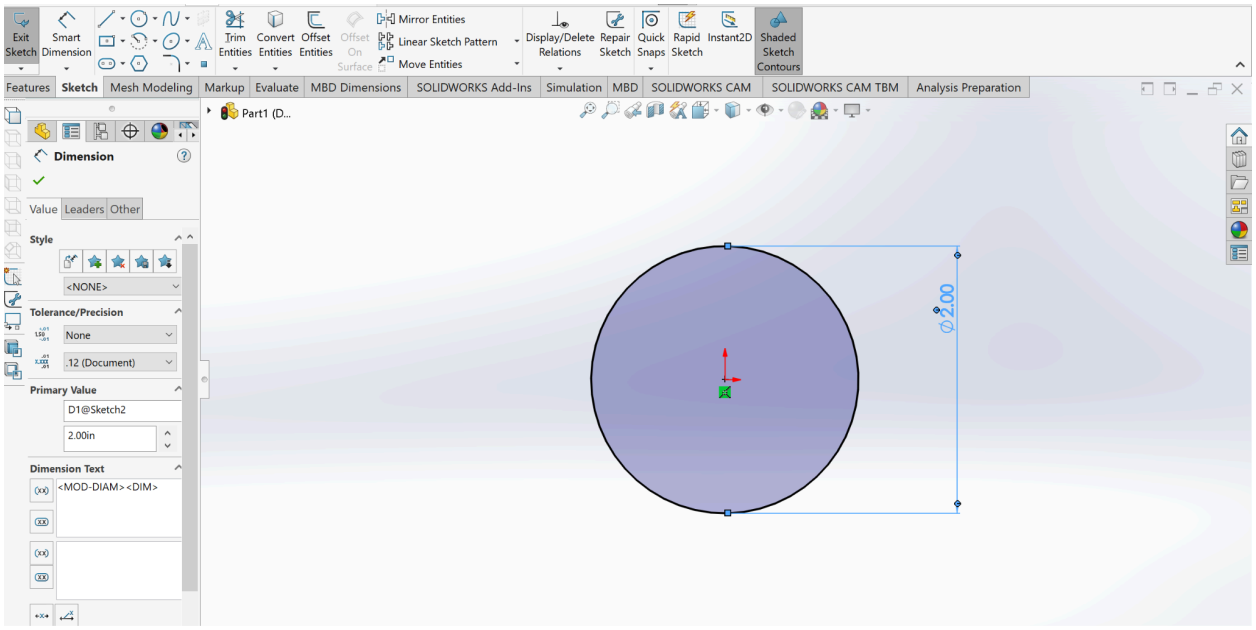
Now let's make something a little more complex. In this example, we are going to make a wheel. First step is to delete the block that we have created (sorry). Right click where it says Boss-Extrude1 in the directory on the left. You should be left with sketch1, and then do the same action towards the sketch. This will give you a clean sketch. See image to the left.



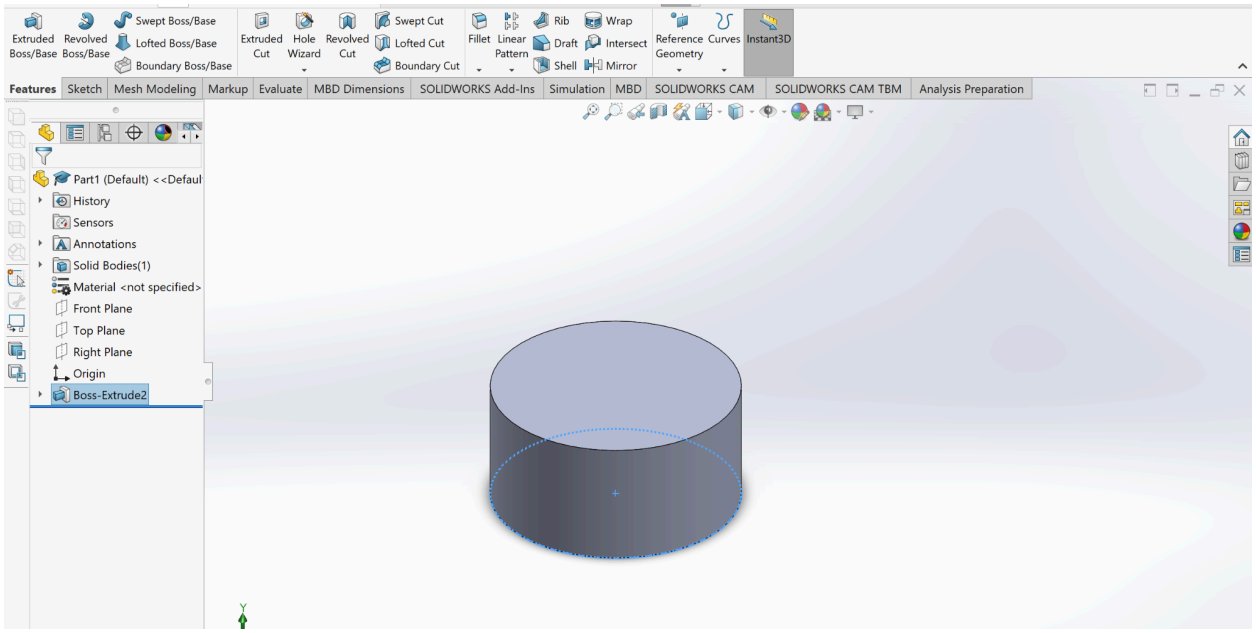
With the clean sketch, let's make a cylinder. The first step is to again click the sketch tab on the top ribbon and then click the sketch command. Instead of selecting the rectangle command, select the circle command which is directly to the right of the line command. Clicking the dropdown gives two options in creating the circle: circle and perimeter circle. Circle allows you to create a circle by first clicking the location of its center, and by moving the mouse expands its size. Clicking again allows you to create the shape. Perimeter is similar except you select three points on the edge of the circle to create it. For now, let's use the circle command. Click the circle command and select the center to be at the origin. Then expand the circle and click again. It should look like something below.



We can also Smart Dimension the circle as we did with the rectangle. Let's make it 2 inches. It should look like below.

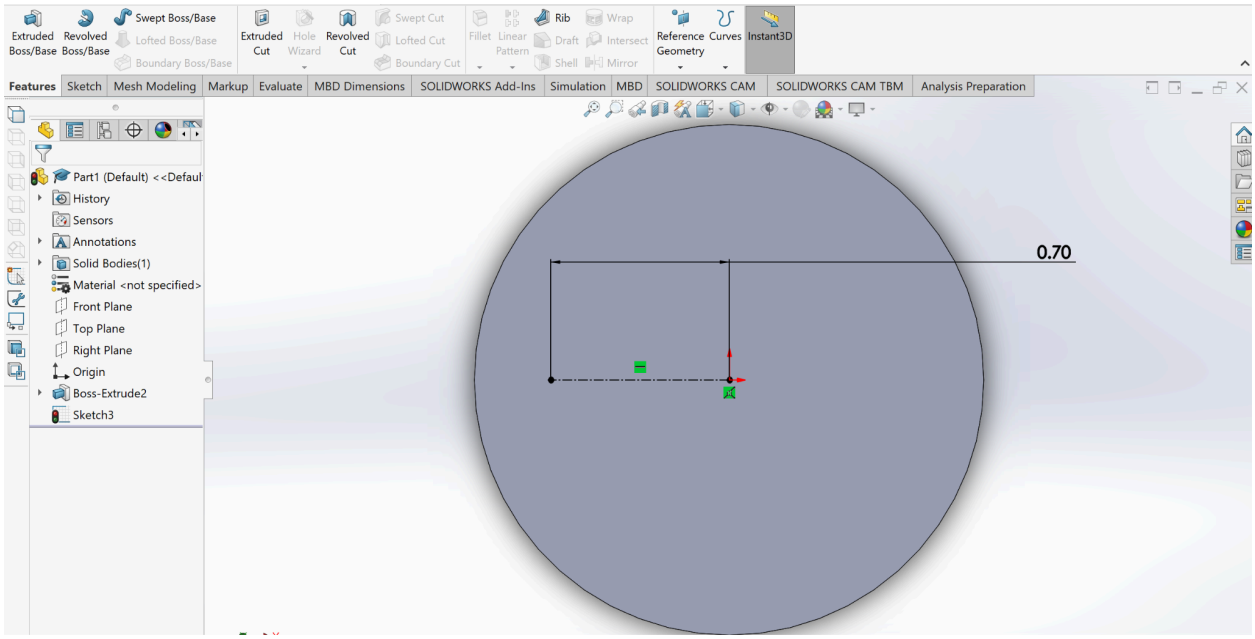


Now let's extrude it 1 inch. Same process with the rectangle and should look like this:

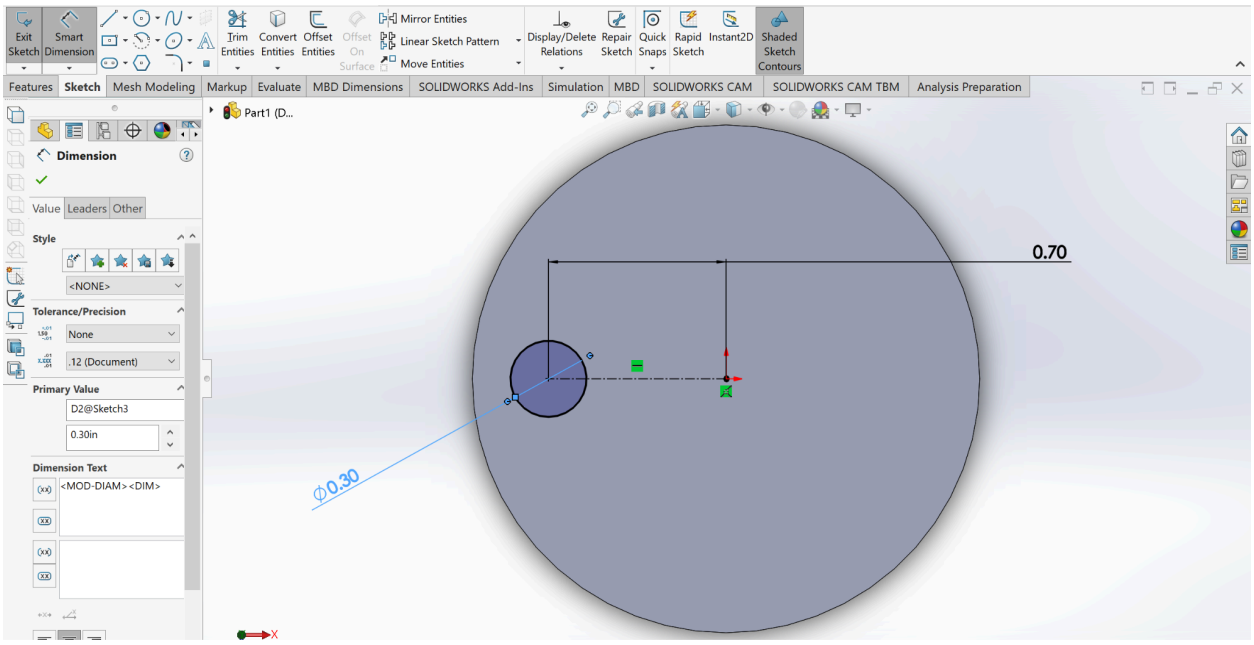


We've done sketching on the planes, but did you know we can sketch on our existing models as well? That is what we are going to do now. To sketch on it , click the surface you want to sketch on, which will be the top of the cylinder, then click sketch. We are going to add holes

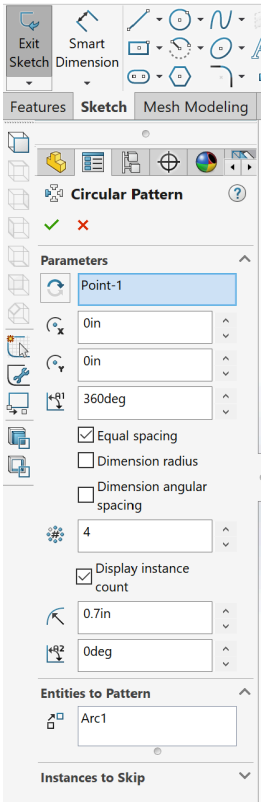
in a circular pattern around on top of the cylinder. Let's use the centerline to help give us a place to click on for the center of these smaller circles. Click the line command dropdown and click on the 2nd option, then click the origin and extend 0.7 to the left. Smart Dimension to make it 0.7 inches. The Smart Dimension tool is in the sketch ribbon, and right next to the 'sketch' command.



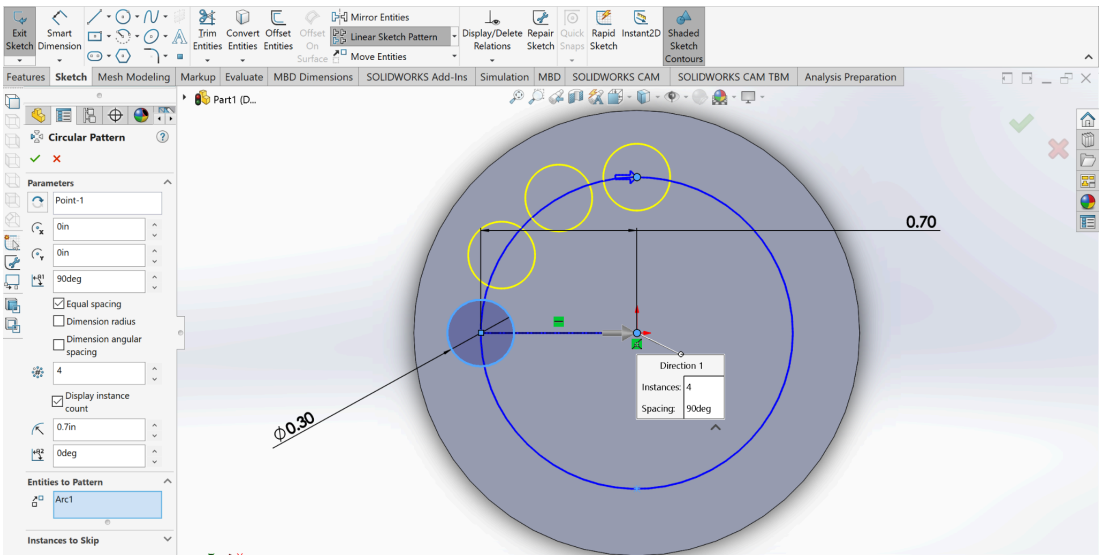
Next, click the circle dropdown and click circle. Use your first click to center the circle at the end of the line. Extend the circle outwards and then use the Smart Dimension command to make the circle 0.3 inches in diameter.



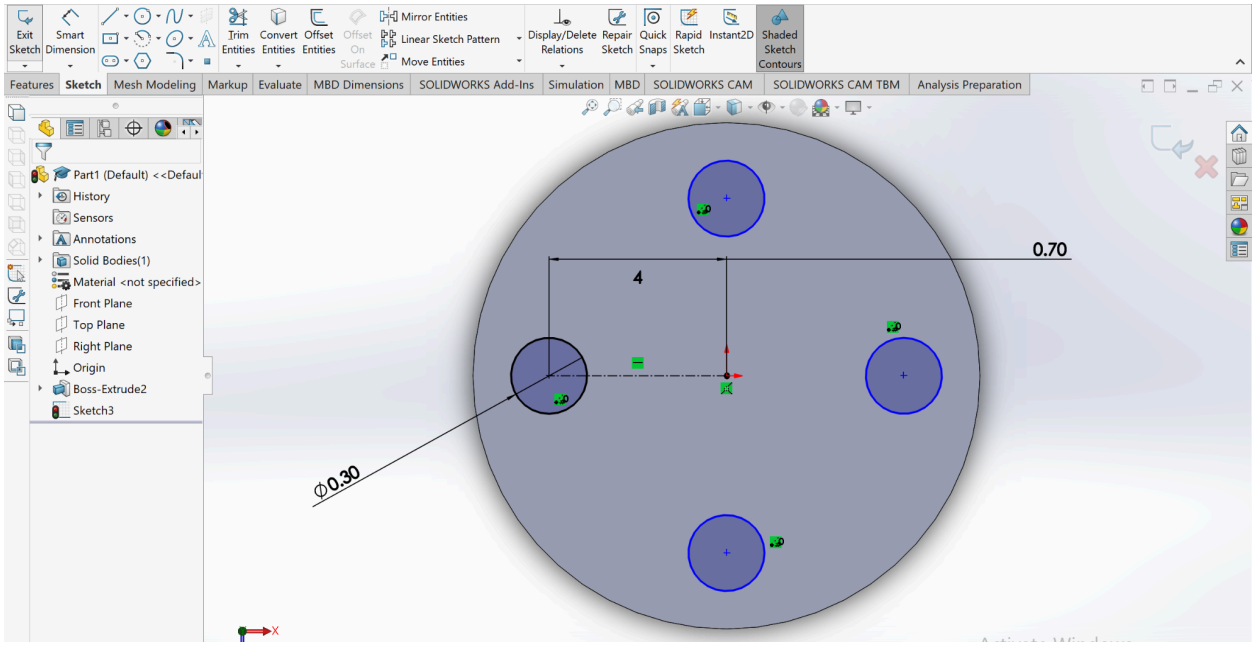
Now we will introduce a new feature: sketch patterns. Moving on the right of the ribbon in the sketch menu, next to Offset on Surface, it will be listed as the linear sketch pattern. Click the dropdown and you have linear or circular sketch pattern options. Linear allows you to repeat your sketch linearly along a line spaced at a specific distance. Circular does the same thing but just in a circle, separating by degrees. We are going to use the circular one. Click the circular sketch pattern in the dropdown. Your first click will be the sketch you want to repeat.



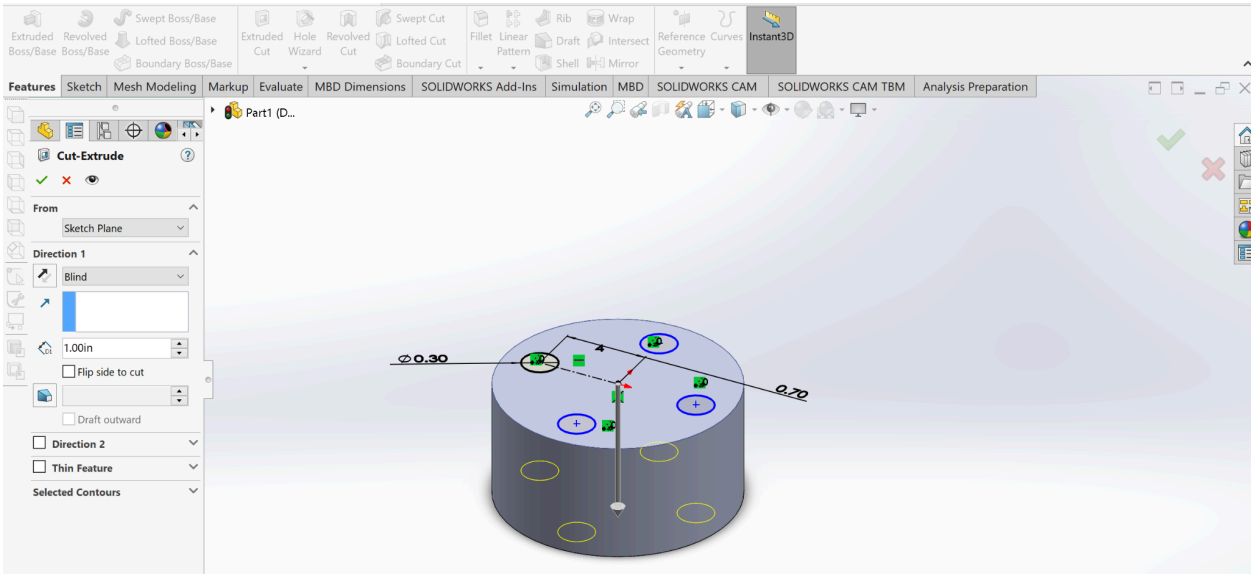
There are a lot of options on the directory on the left of the screen. We are only going to mess with a few of them. The equal spacing check box is important if you want your pattern to be equally spaced. It's convenient because below it there is a number four, which represents how many repetitions of the patterns you want. By checking the equally spaced box, it spaces them out equally for you. Below the Y there is an angle command. Changing that will change how far the pattern will go. For instance, we have 360 degrees indicated, which means the pattern will go around a complete 360 degrees. If we had 90 it would only go to 90 degrees. Here is an example of the 90:

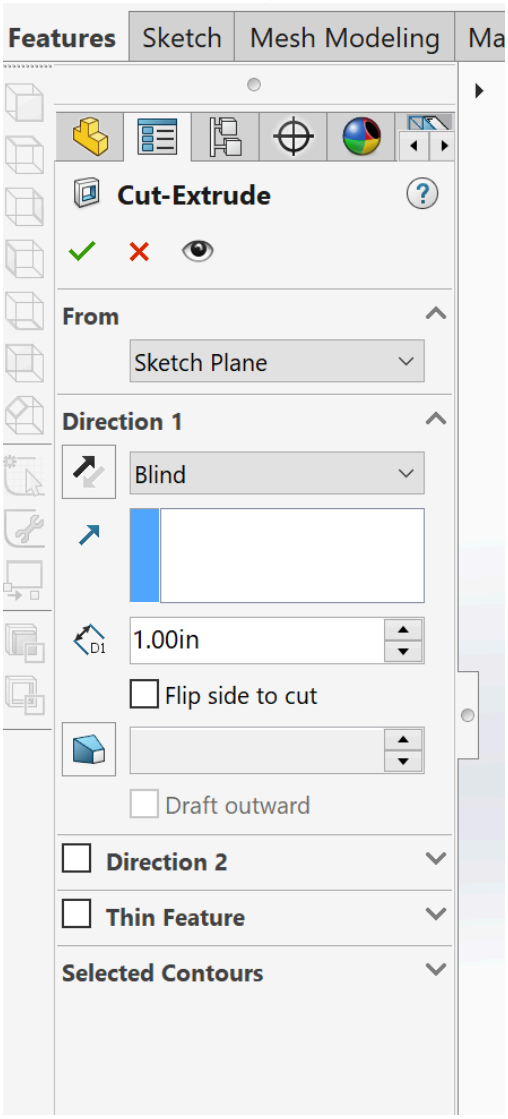


Here is the 360 which we want:



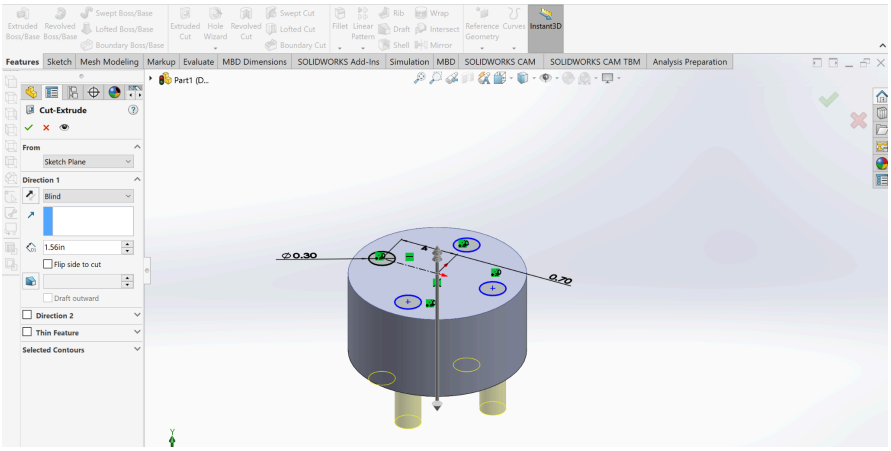
Next, let's make some holes with these smaller circles. We are introducing the extrude cut function. Click on the features tab in the ribbon while still in sketch and locate 'Extrude cut' to the right of 'extrude boss'. 'Extrude cut' cuts through your solid body based on the shape you indicate. For instance in this example, we are going to cut circles through the larger circle. By clicking the extrude cut command and indicating that our smaller circle sketch is what we want to cut by selecting the four small circles, it should look like this:



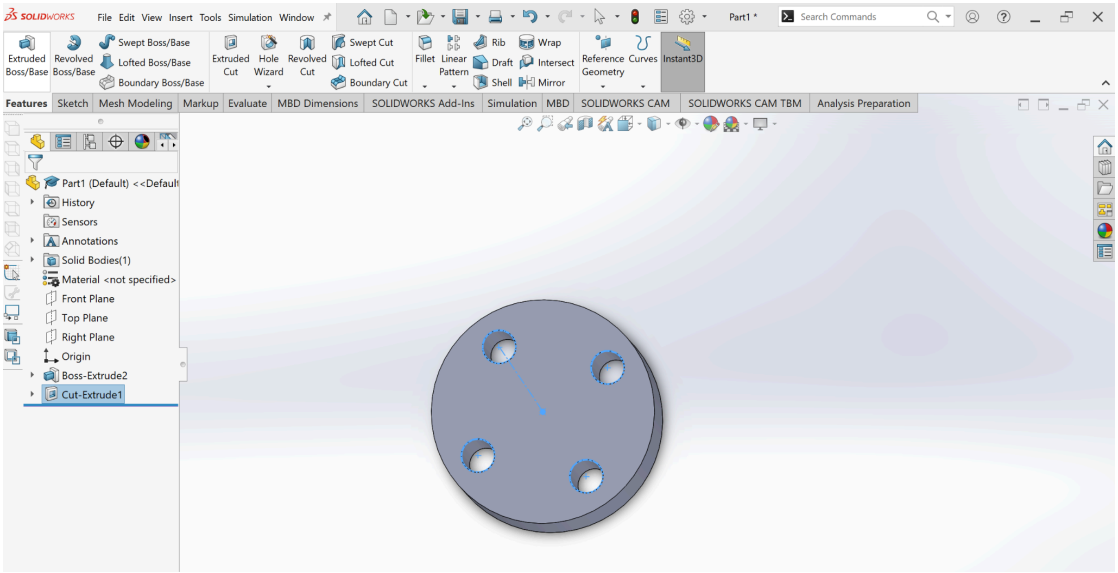


The yellow shows what we are going to cut. This has a similar directory to extrude base and has the same functions as seen in the image on the left.

For instance, if you have 'blind' selected you can drag the extrude and control its distance, similarly to extrude boss base. In this case, let's keep using the blind option. To make sure you cut all the way through, I like to pull the cut all the way through until it is much over the size of the solid body. Drag the arrow freely and it should look like the image below:



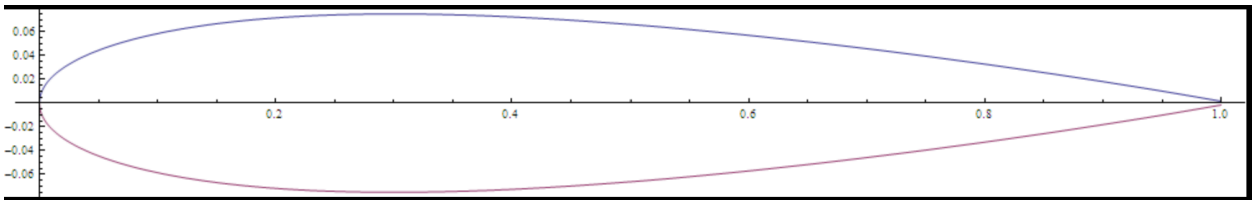
If you are satisfied, click the green checkmark. Now it should look like this:



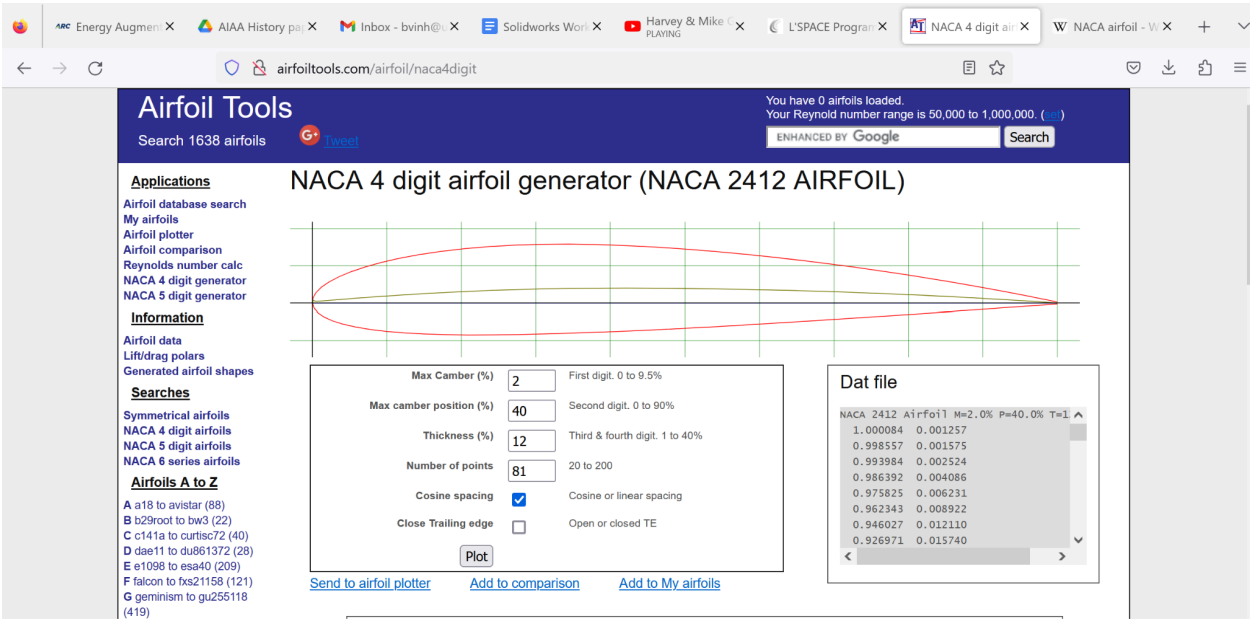
Congratulations! You have made the button.

For the final example, we are going to make a 3D model of a wing. We will introduce a few more new features throughout this example.

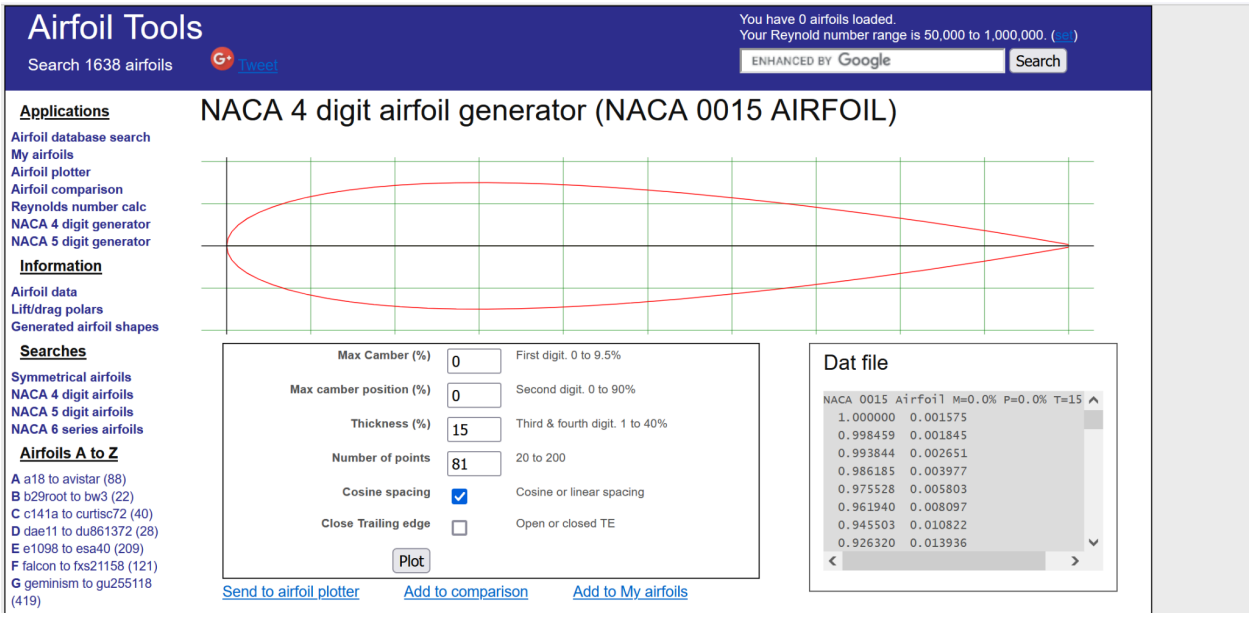
The first step is to head to a website called airfoil tools. Airfoil tools is a large database of airfoils that we can use to gather DAT files, which are x-y coordinates that when drawn out, create the airfoil shape. You can even create custom airfoils if you know the NACA numbers for them. You may be wondering what an airfoil is. An airfoil is essentially the wing shape. The way they are named is by a four digit code, an example is the 0015 NACA airfoil. The first digit means the maximum camber as percentage of the chord. The second digit means the distance of the maximum camber from the airfoil leading edge in tenths of the chord. The last two digits mean maximum thickness of the airfoil as a percentage of the chord. For context, the chord means the distance from the leading edge (front of the airfoil) to the trailing edge (back of the airfoil). Below is an example of a 0015 airfoil, 0 camber, 0 distance of maximum camber, and the maximum thickness is 15 % of the airfoil.



Lets create a wing using this airfoil. You will need the DAT file with the coordinates to sketch the airfoil. To generate the airfoil that we want, click on 'NACA 4 digit generator' underneath applications on the left hand side. You will reach this page below:

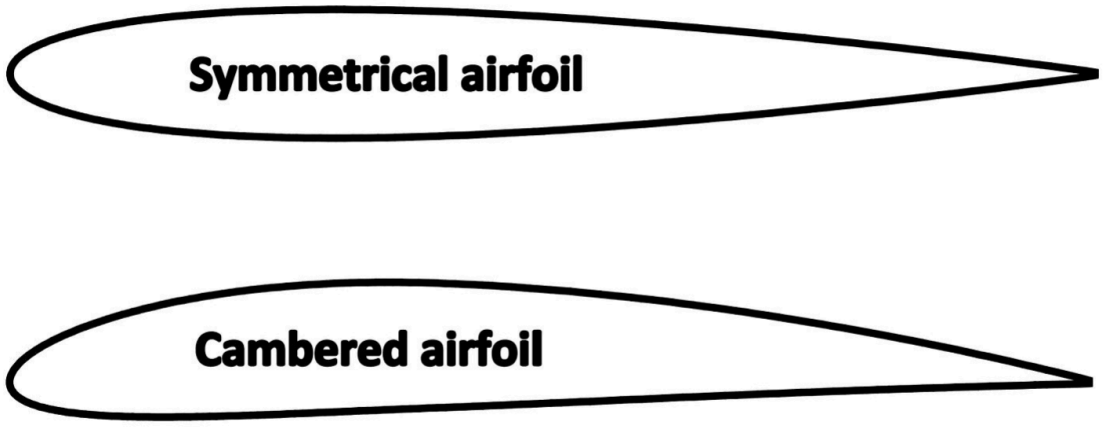


Now we will change the numbers listed in the boxes to fit 0015. The max camber will be 0, the max camber position will be 0 percent, and the thickness will be 15. The number of points does not need to be changed. Then, hit enter and it should look like this:

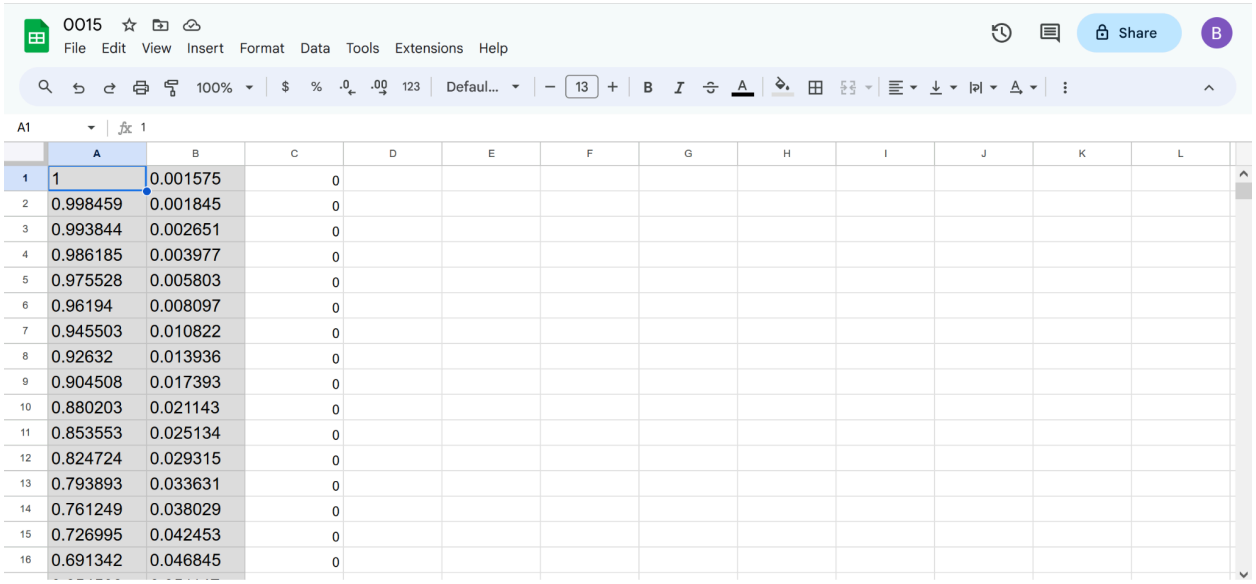


For context airfoils that start with 00 are called symmetric airfoils as the top and bottom of the airfoil is the same shape.

Here is an image of the difference between cambered and symmetric airfoils

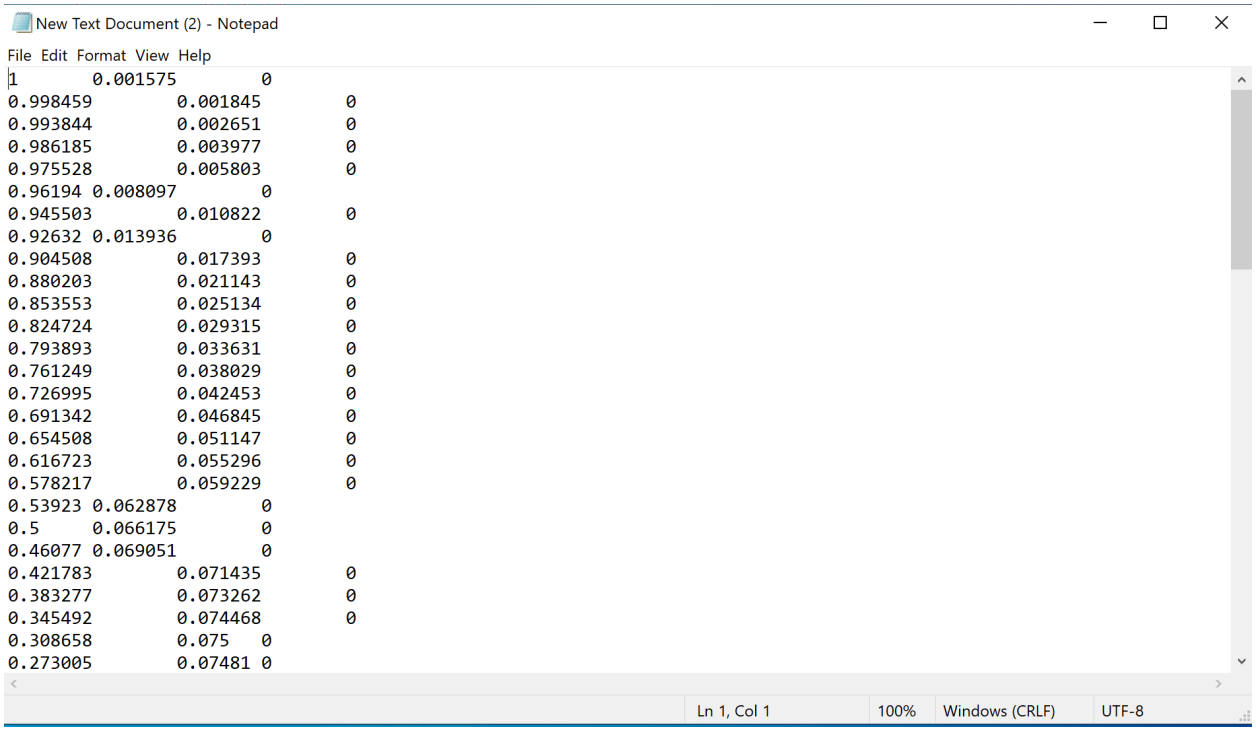


Let's now copy the DAT file located on the right of the digits you just adjusted. Copy everything except the first line and paste that into a google sheets. Once you do that, you need to separate the data into two columns. To do this, click on data in the top ribbon and click separate text to columns. Make sure you have the entire column highlighted. Finally, we need to add a third column of just 0s because Solidworks only takes it if it has three coordinate points:

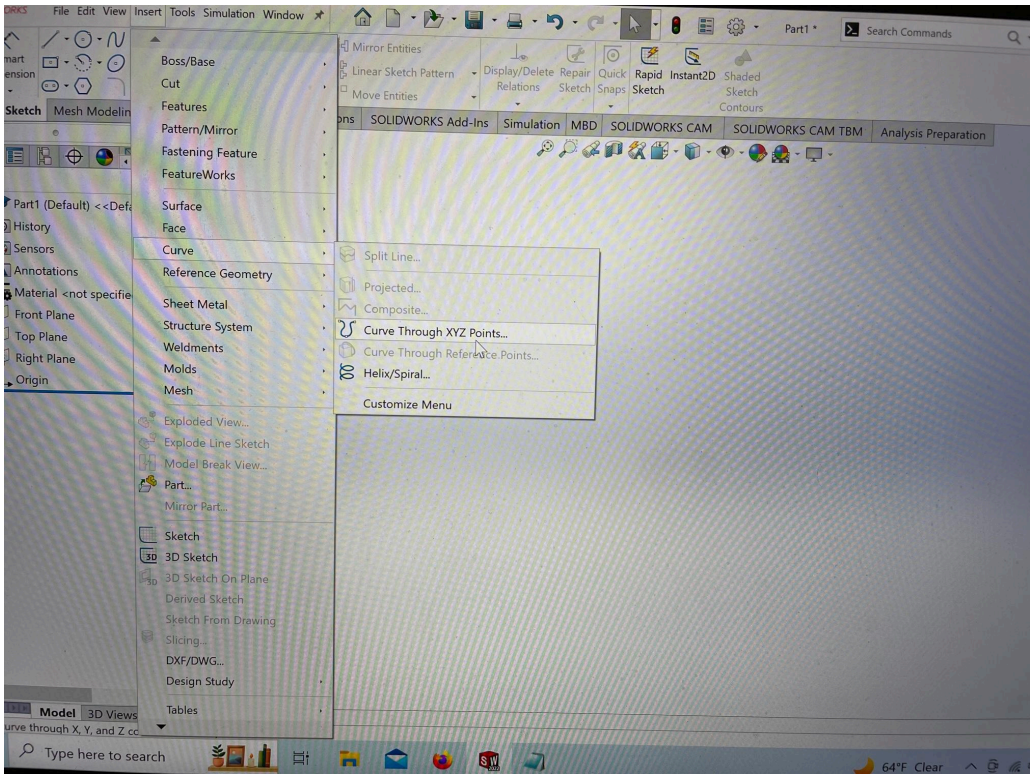


	A	B	C
1	1	0.001575	0
2	0.998459	0.001845	0
3	0.993844	0.002651	0
4	0.986185	0.003977	0
5	0.975528	0.005803	0
6	0.96194	0.008097	0
7	0.945503	0.010822	0
8	0.92632	0.013936	0
9	0.904508	0.017393	0
10	0.880203	0.021143	0
11	0.853553	0.025134	0
12	0.824724	0.029315	0
13	0.793893	0.033631	0
14	0.761249	0.038029	0
15	0.726995	0.042453	0
16	0.691342	0.046845	0

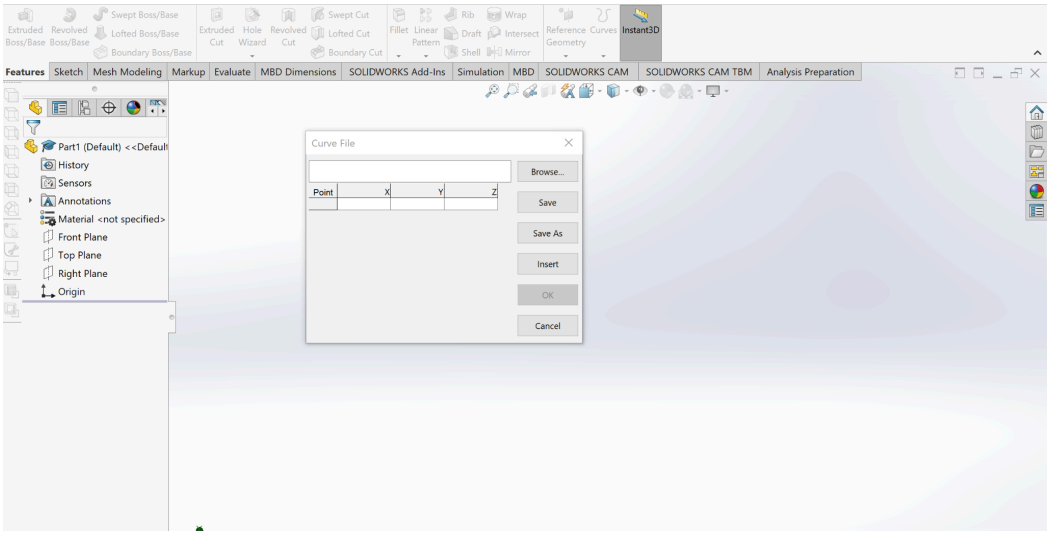
Once you have that done, open a .txt file or a text file and copy the three columns into the .txt file and save it.:



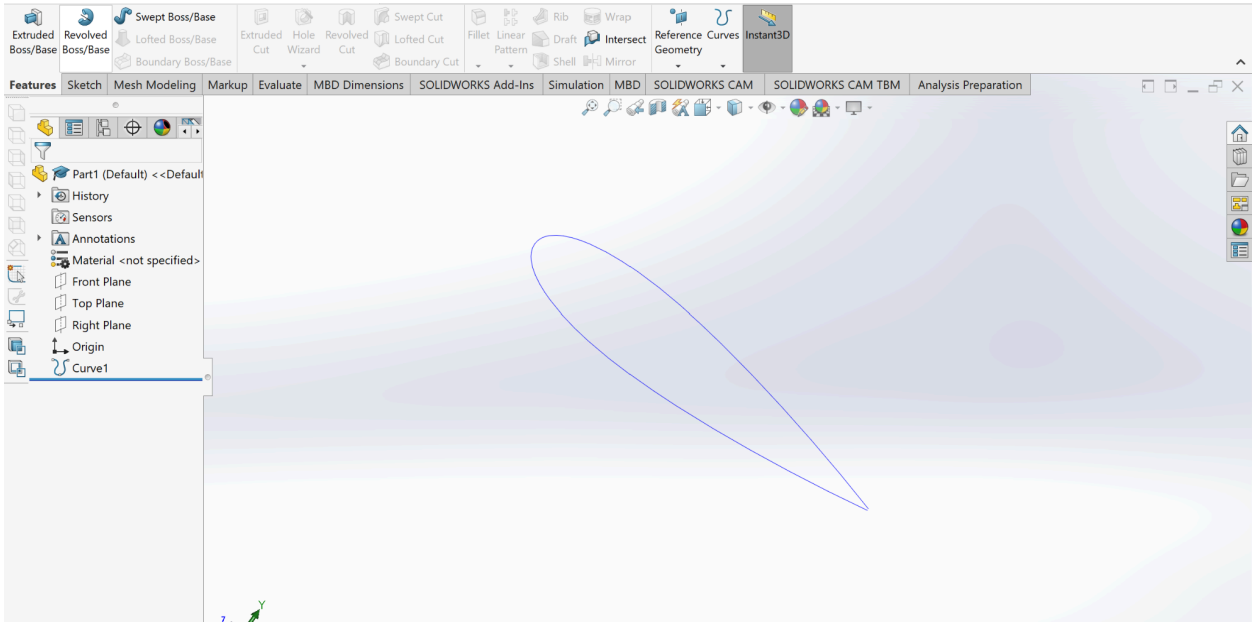
Open solidworks and click ‘Insert’ in the top ribbon, then ‘curve’, the ‘curve through XYZ points’.



You should end up here:



Hit browse and open the .txt file with the coordinates you just inputted. It should show up onto the graph and the airfoil will appear as a sketch on the screen. If you did it right, it looks like this:

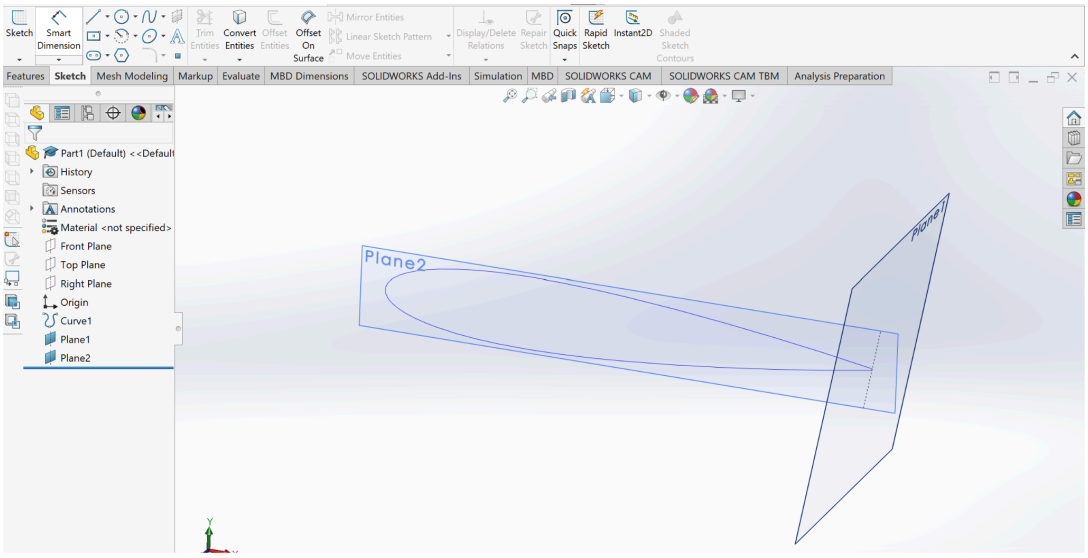


We can't manipulate this because it is just a curve, not an actual sketch that we drew. We need to project this onto a reference plane. Click on features in the ribbon and locate

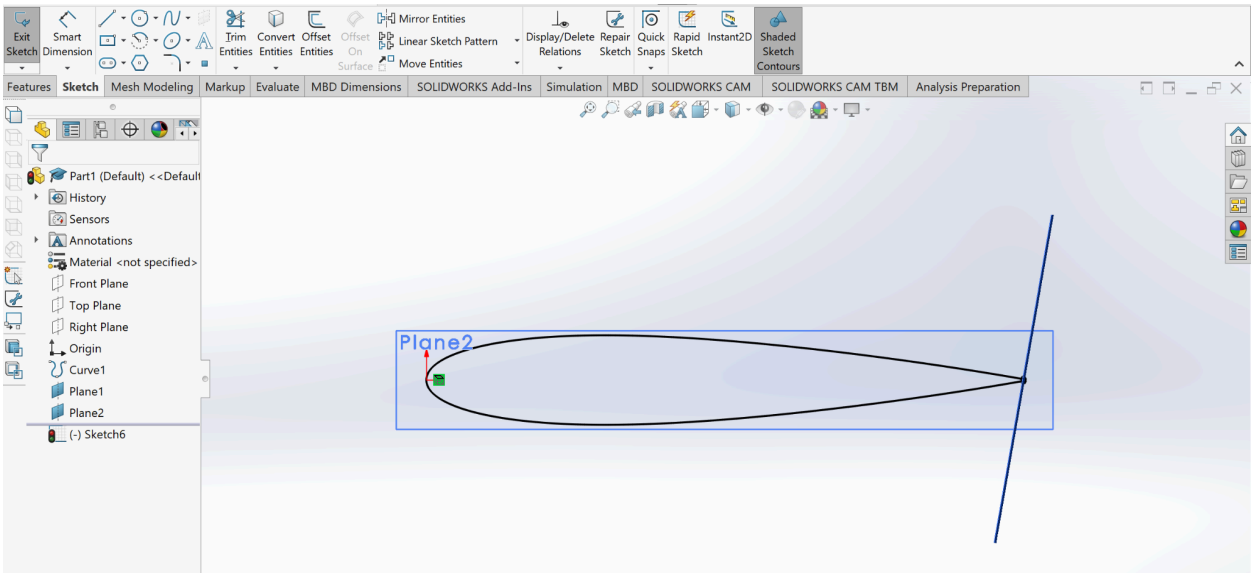
reference geometry near the end of the feature list on the right.

When you click on reference geometry, a drop down appears.

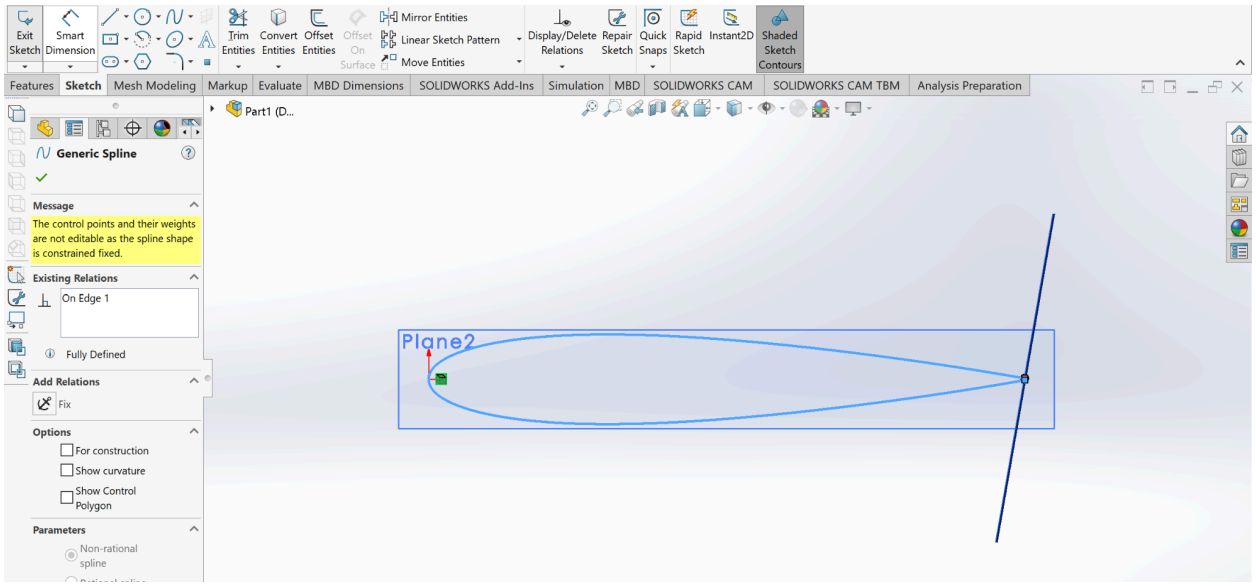
These are options for a geometry that you can place in reference to whatever you need it to be. For now, click on plane to create a reference plane. For your reference point, click the sketch of the airfoil. The plane appearing should look like this:



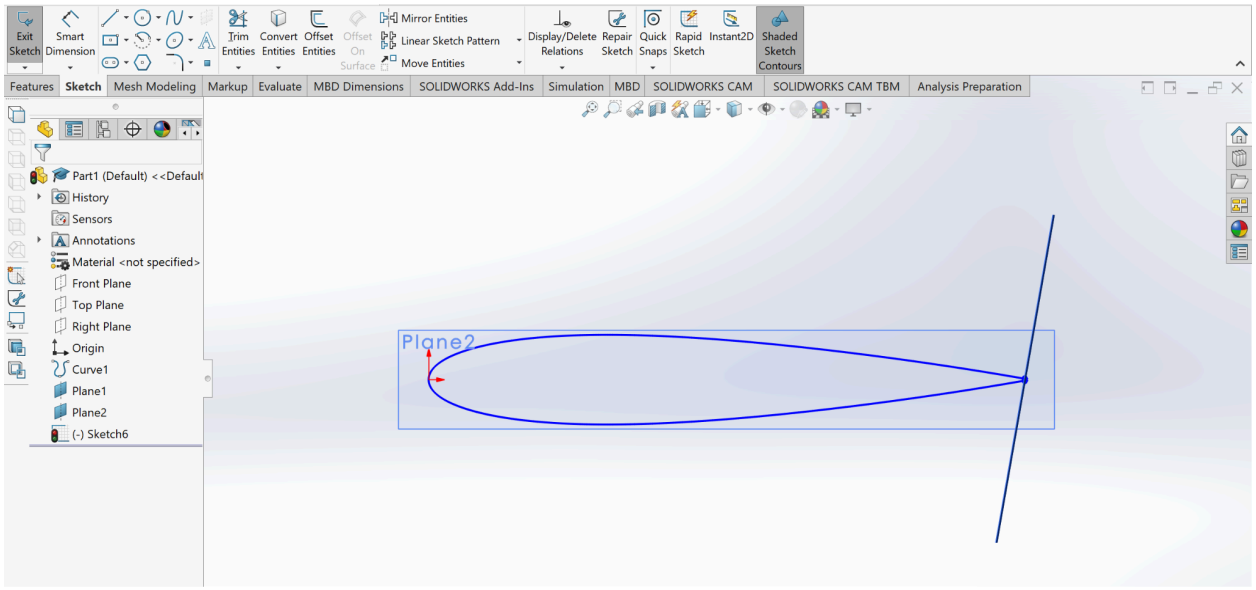
Now we need to move the curve onto the plane so that we can manipulate it. To do this, click the plane (in this case it is labeled Plane 2) and then click sketch. Then click on the Convert Entities command, which converts the sketch of another plane and moves it to the plane you are sketching on. It will look like this



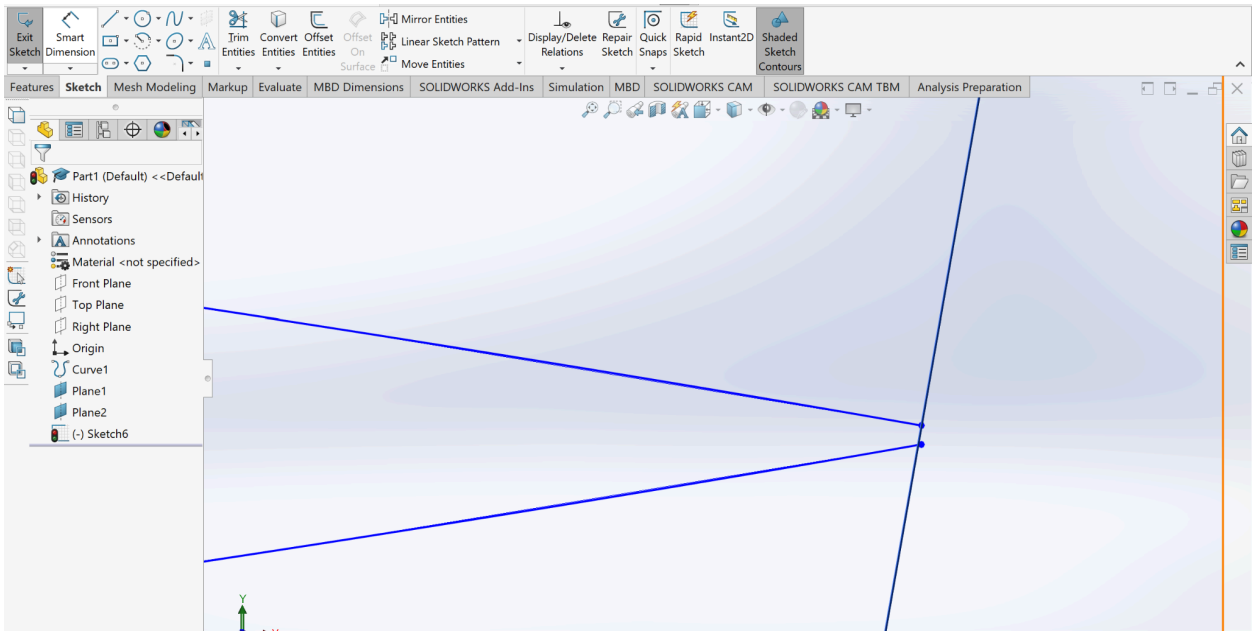
Notice how the sketch is black. That means it has a relation that doesn't allow us to mess with it. To remove this, click the black line, and on the left hand side it will display a relation called on-edge. On-edge means that the sketch is defined on its edge onto the plane so it's unmovable.



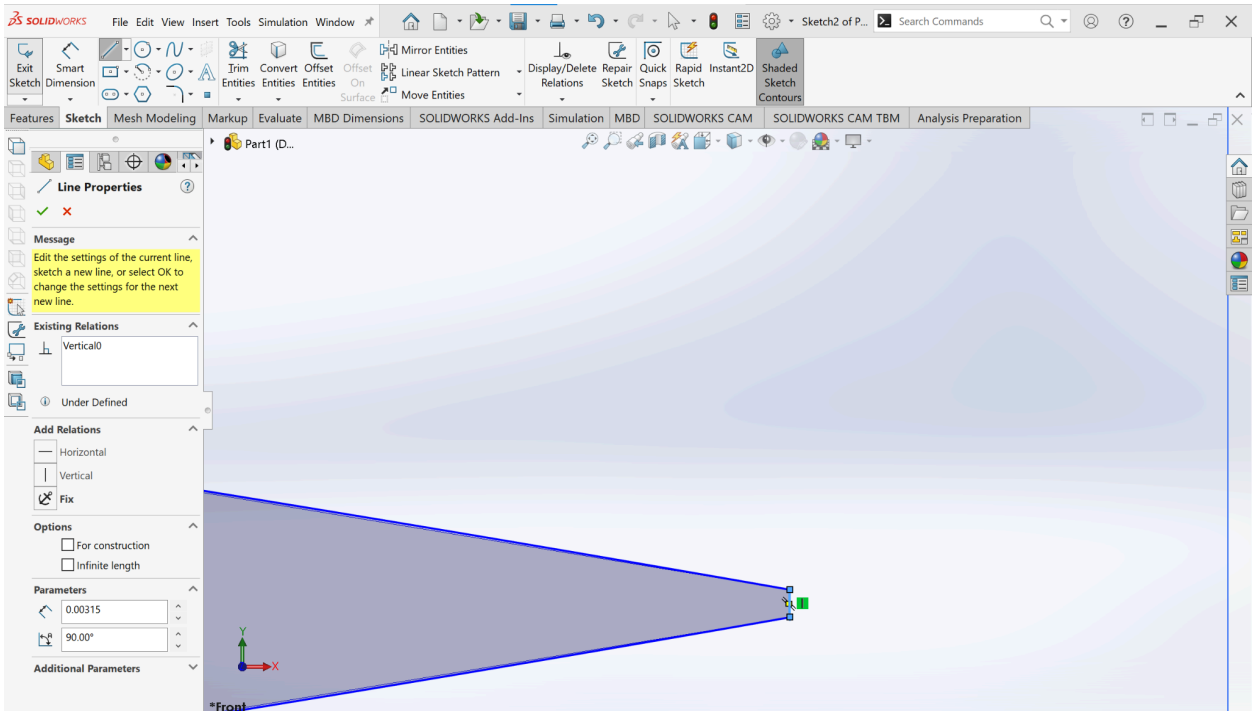
Delete the on-Edge relation by right clicking it and hitting delete. The sketch should turn blue.

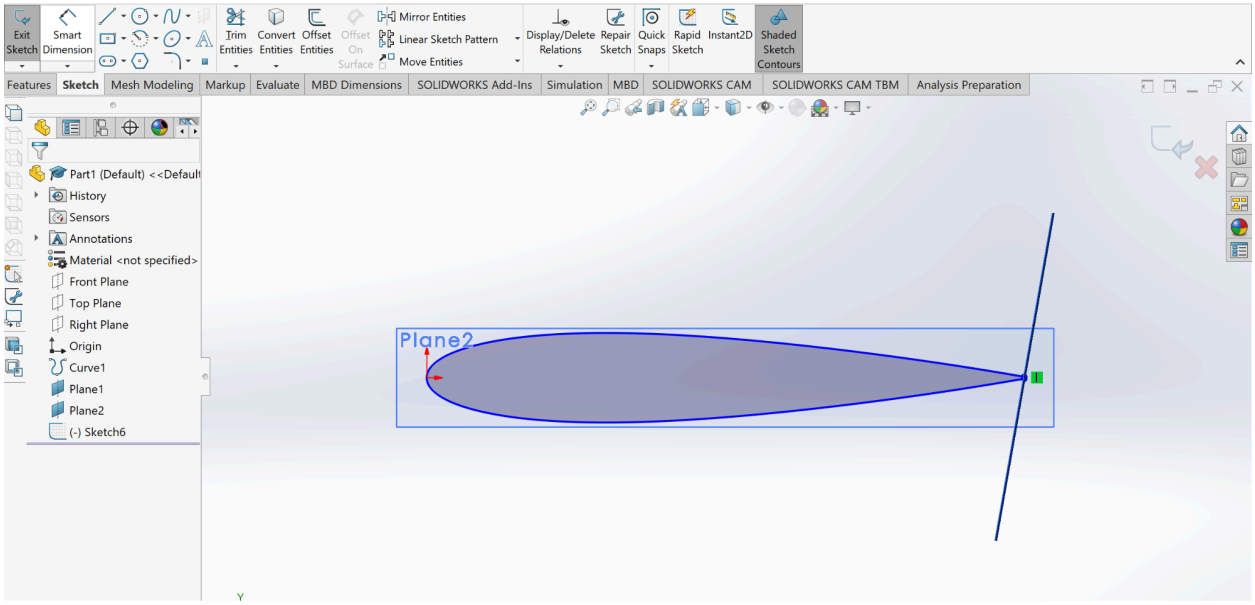


But wait. We can't have a wing if it is not 3D. The issue is that if a sketch is closed, the interior of the sketch would be greyish. In this case, it is not. The sketch is opened. Zoom in on the trailing edge of the airfoil



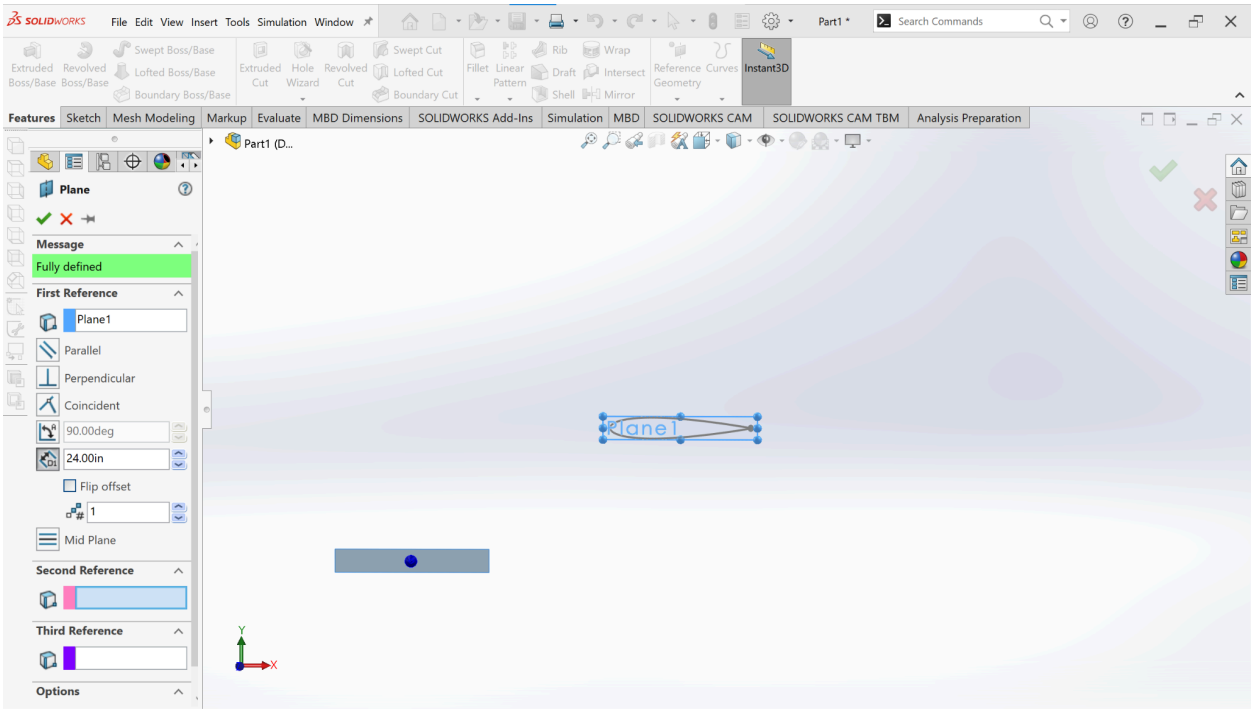
We need to close that gap, all you need to do is sketch a line to close it. Make sure you are in sketch mode and click on the 'line' command. Then click the bottom dot at the trailing edge and connect it to the top dot.

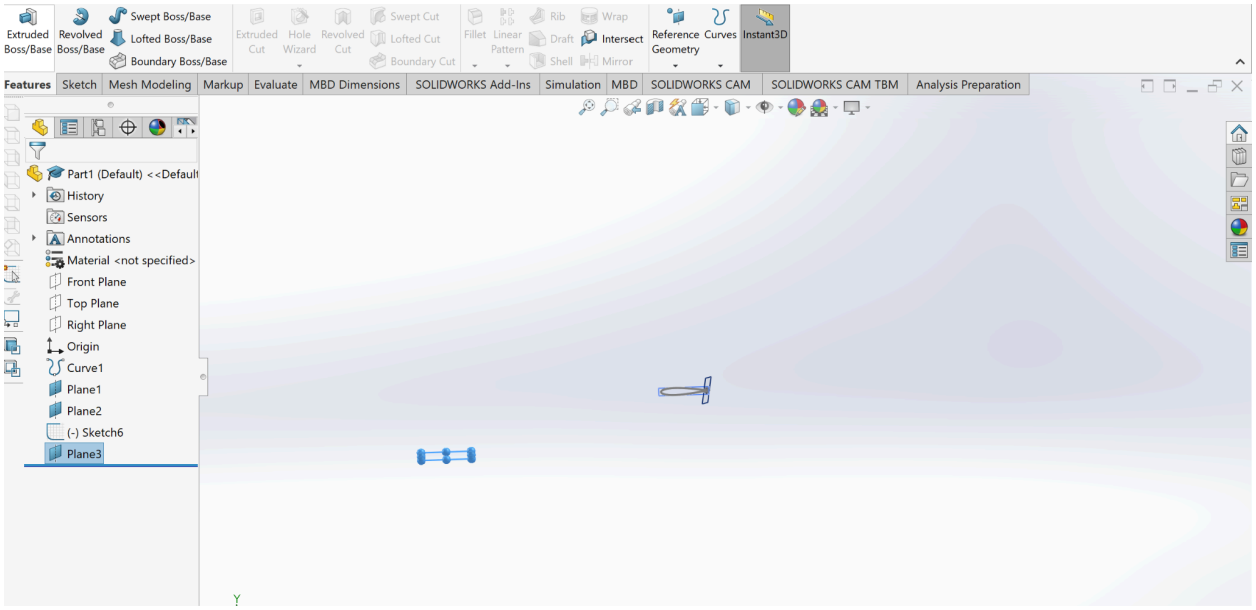




Now it is closed, exit the sketch.

Let's add a small taper to the wing to make it look better. To do this, make another reference plane about 2 feet from the original plane. What you would need to do is go to the 'features' ribbon, then click on 'reference geometry' to the right of the feature list, and click on plane from the drop down. Then select Plane 1 as your reference and type in 24 inches for distance in the distance box. See the right directory:

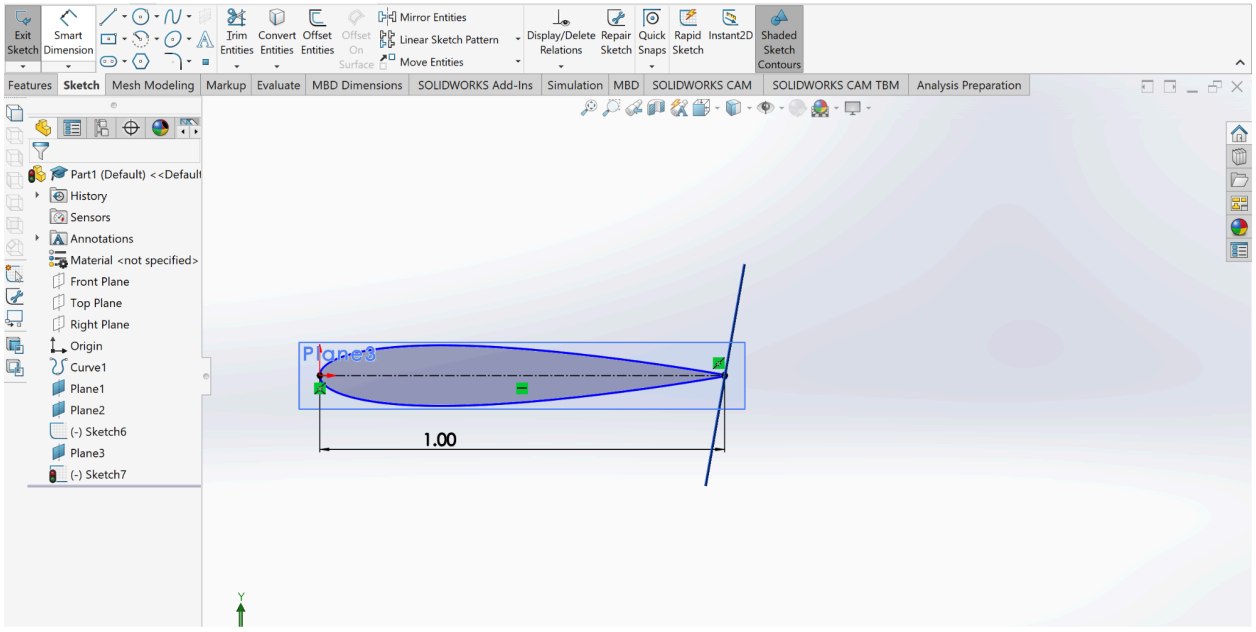




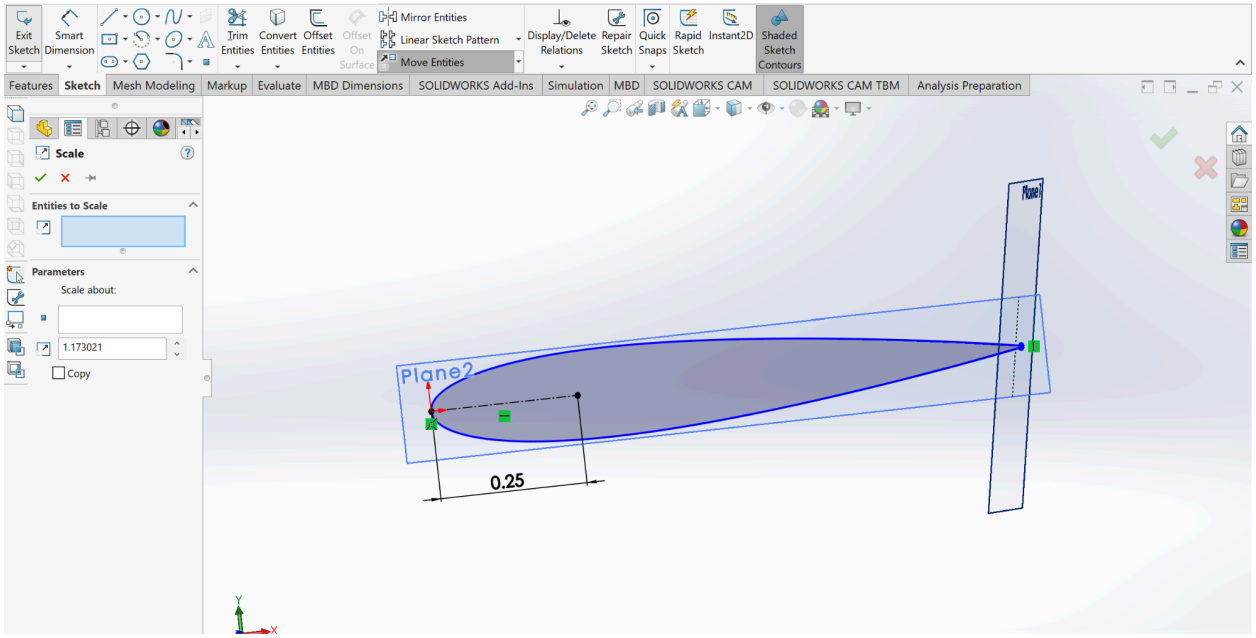
Next, click on the new plane and hit sketch. Then convert entities of the airfoil onto the new plane. Make sure you zoom in onto the trailing edge and click the line you drew as well to make sure it gets converted as well.

The sketch will appear as a black line again, this means you need to delete the relation. Delete the relation for both the curve and the line you drew on the trailing edge. Now, the entire sketch should be blue.

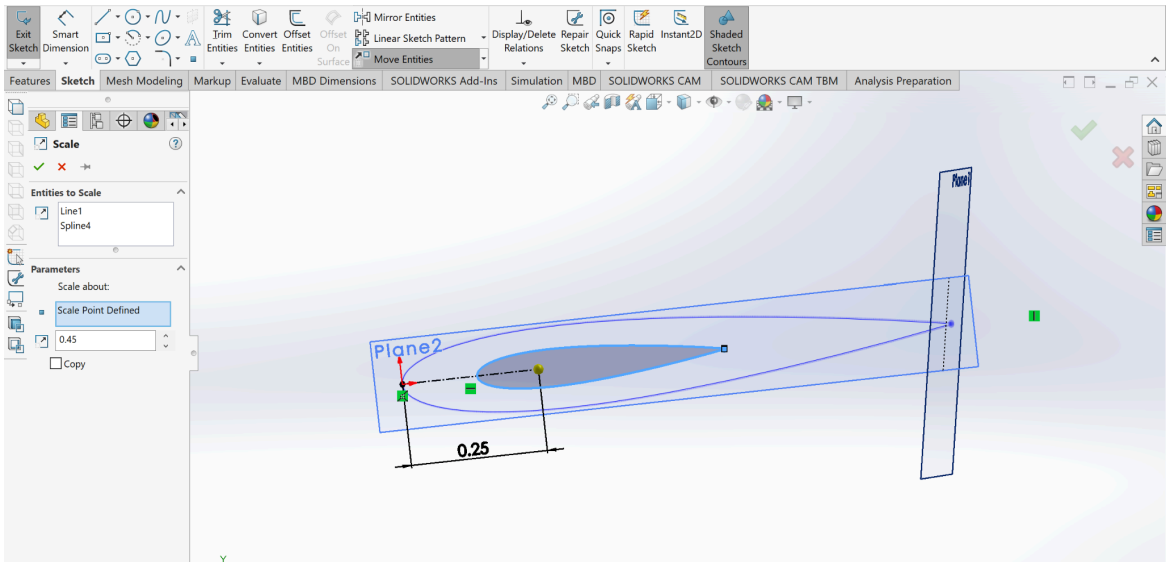
To create the taper, we need to make the tip of the wing or the tip chord, smaller than the root chord. We need to keep the aerodynamic centres of both of the airfoils straight on each other. In other words, if we draw a straight from the root aerodynamic centre, it should line up to the tip chord aerodynamic centre. The aerodynamic centre is located a quarter of the chord length starting from the leading edge. In this case, let's draw a construction line from the middle of the leading edge to the middle of the trailing edge to see the total chord length.



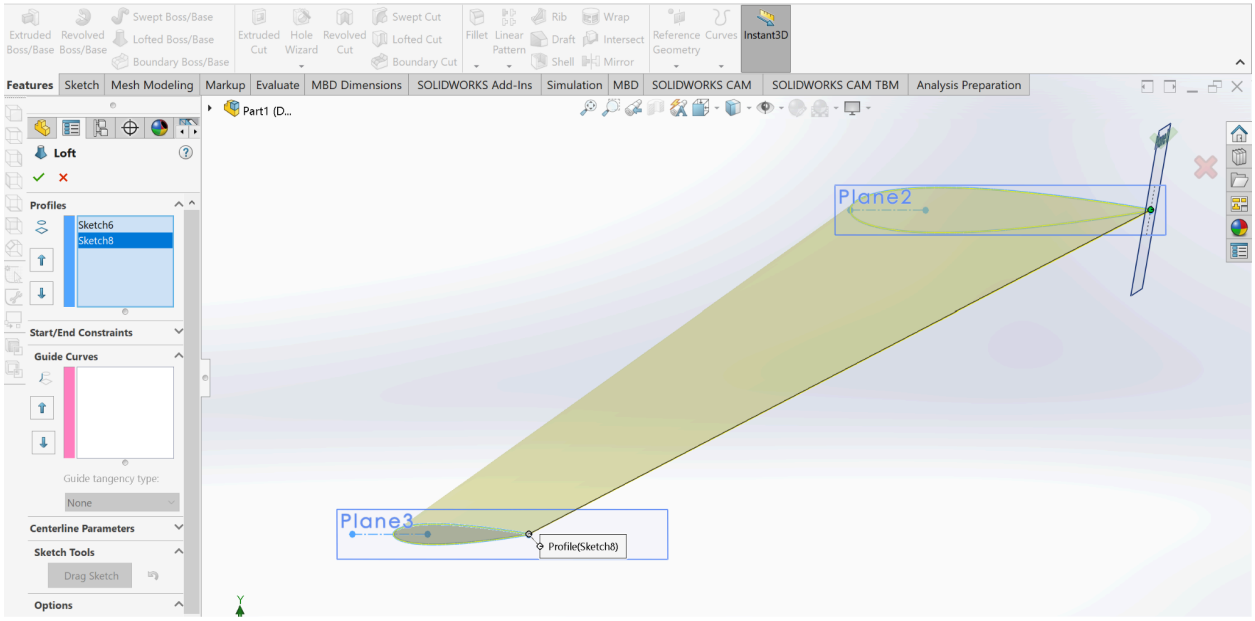
As you can see, it is 1 inch, therefore the aerodynamic centre is 0.25 inches from the leading edge. Delete the line you drew and redraw it starting from the leading edge to the length of a quarter of the chord length, which in this case is 0.25 inches. Next, type in the search bar scale and click on scale entities.



This will allow you to scale your sketches up or down and at a specific scaling point you indicate. Right now, the first click is on what entities we want to scale. You can tell because on the left the entities to scale box is blue meaning we are selecting for that part. Click on the curve and the line you drew at the trailing edge. Then click on the 'scale about' box and click on the aerodynamic center. Next, click on the box right below it with numbers, which is your scale factor. Change it to 0.45, or 45 percent of the original length. This will give us a taper of 0.45.



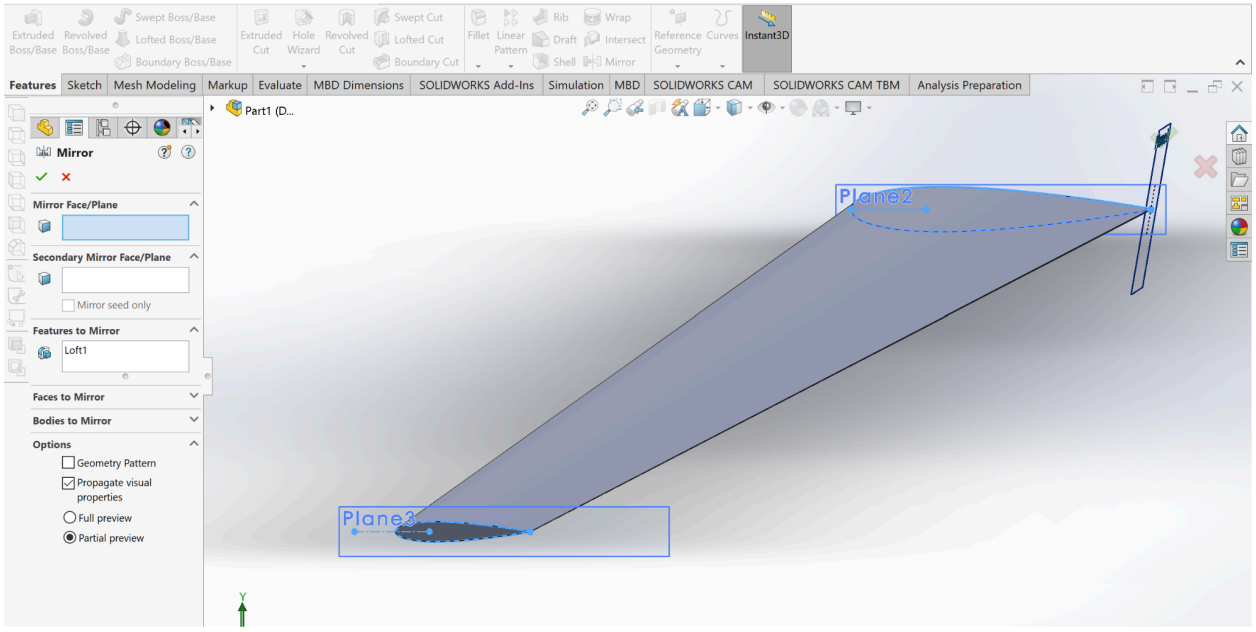
Click the green check mark if it looks like the image above. Finally, we are going to make the body come to life. Exit the sketch and click on the feature tab in the ribbon. Now click on Lofted Boss/base which is located to the right of revolved boss/base. Lofted Boss/Base creates a solid body linking two sketches together. Once you click on the lofted boss/base command, it will ask you to select your profiles. Select the larger airfoil first and then the smaller one. It should give you a preview.



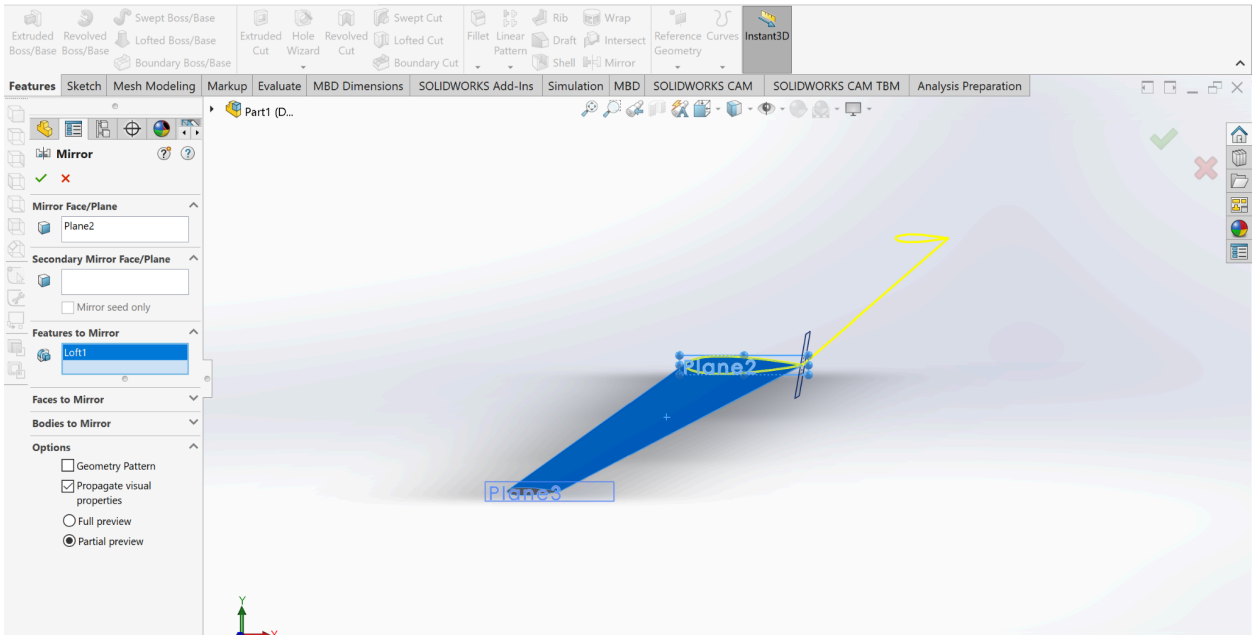
If it looks like this, hit the green checkmark and it should appear as a solid, grey body.

It looks to me like we have made half a wing. We need to make the other half, but there is a faster way than completing the same steps again.

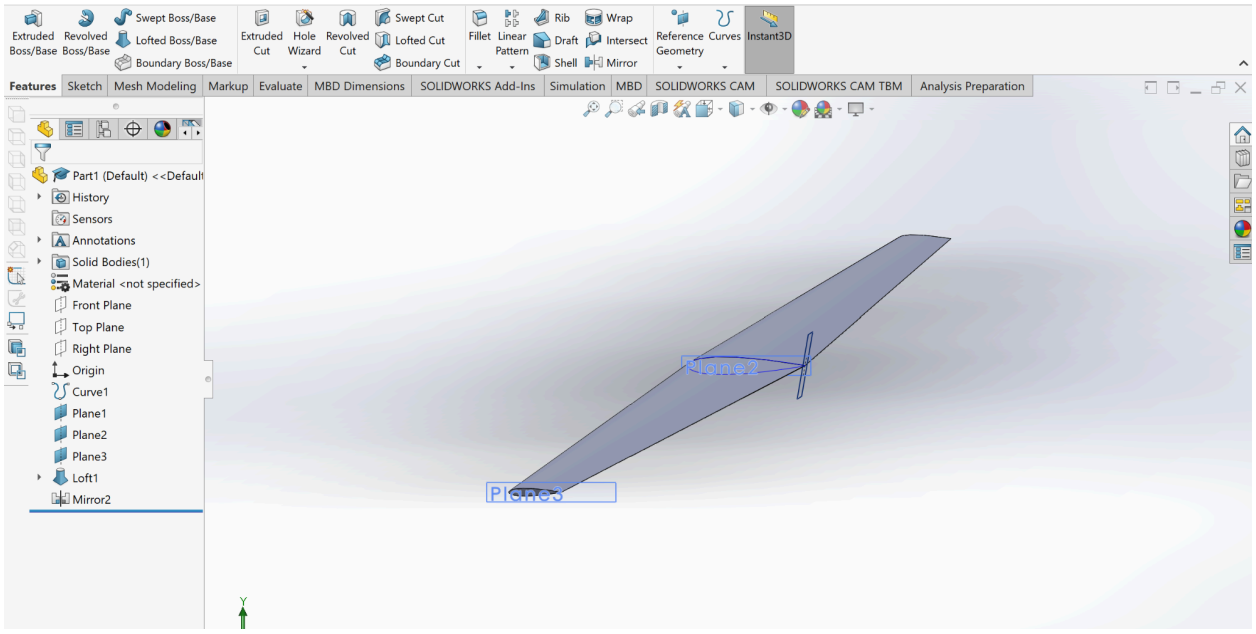
Go to the search bar again and type in mirror and hit entre. Click on the command that says mirror.



The blue box is indicating that we need to select the mirror face/plane first. This is the place/face for which we are going to mirror over. Click on plane 2 for that. Next select the features to mirror box and click the loft you've created. It should look like this:



Once you are satisfied, click the green checkmark. If everything has gone as planned it should look like this:



Congratulations! You have made the wing and successfully completed the tutorial! Play around with Solidworks more, the more practice you get the better.